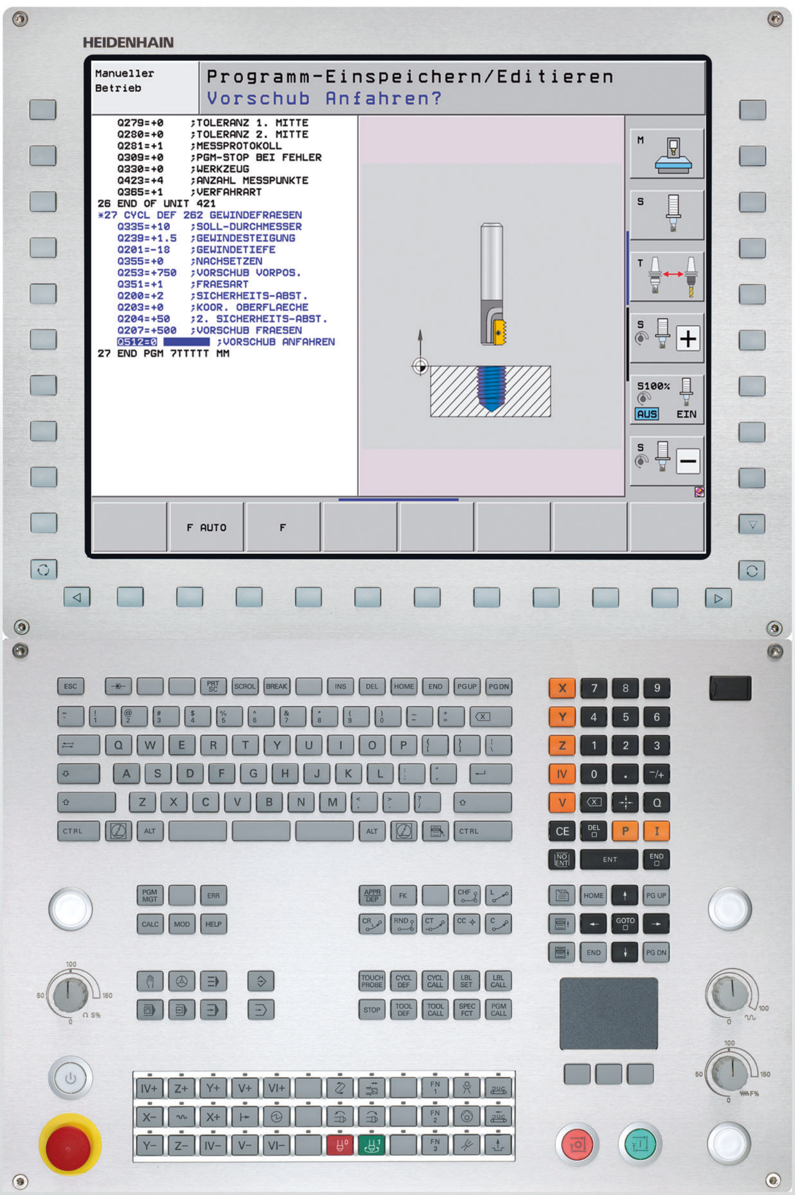
HEIDENHAIN



User's Manual
Cycle Programming

iTNC 530

NC Software
606420-04, SP8
606421-04, SP8
606424-04, SP8

English (en)
3/2016

About this manual

The symbols used in this manual are described below.

This symbol indicates that important information about the function described must be considered.

This symbol indicates that there is one or more of the following risks when using the described function:

- Danger to workpiece
- Danger to fixtures
- Danger to tool
- Danger to machine
- Danger to operator

This symbol indicates that the described function must be adapted by the machine tool builder. The function described may therefore vary depending on the machine.

This symbol indicates that you can find detailed information about a function in another manual.

Would you like any changes, or have you found any errors?

We are continuously striving to improve our documentation for you. Please help us by sending your requests to the following e-mail address: **tnc-userdoc@heidenhain.de**.

TNC model, software and features

This manual describes functions and features provided by TNCs as of the following NC software numbers.

TNC model	NC software number
iTNC 530, HSCI and HEROS 5	606420-04, SP8
iTNC 530 E, HSCI and HEROS 5	606421-04, SP8
iTNC 530 HSCI Programming Station	606424-04, SP8
iTNC 530 programming station, HEROS 5 for virtualization software	606425-04, SP8

The suffix E indicates the export version of the TNC. The export version of the TNC has the following limitations:

- Simultaneous linear movement in up to 4 axes

HSCI (HEIDENHAIN Serial Controller Interface) identifies the new hardware platform of the TNC controls.

HEROS 5 identifies the new operating system of HSCI-based TNC controls.

The machine tool builder adapts the usable features of the TNC to his machine by setting machine parameters. Some of the functions described in this manual may therefore not be among the features provided by the TNC on your machine tool.

TNC functions that may not be available on your machine include:

- Tool measurement with the TT

Please contact your machine tool builder to become familiar with the features of your machine.

Many machine manufacturers, as well as HEIDENHAIN, offer programming courses for the TNCs. We recommend these courses as an effective way of improving your programming skill and sharing information and ideas with other TNC users.

User's Manual:

All TNC functions not connected to cycles are described in the iTNC 530 User’s Manual. Please contact HEIDENHAIN if you require a copy of this User's Manual.

ID of User's Manual for Conversational Programming: 737759-xx.

ID of User’s Manual for DIN/ISO Programming: 737760-xx.

smarT.NC user documentation:

The smarT.NC operating mode is described in a separate Pilot. Please contact HEIDENHAIN if you require a copy of this Pilot. ID: 533191-xx.

Software options

The iTNC 530 features various software options that can be enabled by you or your machine tool builder. Each option is to be enabled separately and contains the following respective functions:

Software option 1
Cylinder surface interpolation (Cycles 27, 28, 29 and 39)
Feed rate in mm/min for rotary axes: **M116**
Tilting the machining plane (Cycle 19, **PLANE** function and 3-D ROT soft key in the Manual Operation mode)
Circular in 3 axes with tilted working plane

Software option 2
5-axis interpolation
Spline interpolation
3-D machining: ■ **M114**: Automatic compensation of machine geometry when working with tilted axes ■ **M128**: Maintaining the position of the tool tip when positioning with swivel axes (TCPM) ■ **FUNCTION TCPM**: Maintaining the position of the tool tip when positioning with tilted axes (TCPM) in selectable modes ■ **M144**: Compensating the machine's kinematic configuration for ACTUAL/NOMINAL positions at end of block ■ Additional parameters for **finishing/roughing** and **tolerance for rotary axes** in Cycle 32 (G62) ■ **LN** blocks (3-D compensation)

DCM Collision software option	Description
Function that monitors areas defined by the machine manufacturer to prevent collisions.	User's Manual for Conversational Programming

DXF Converter software option	Description
Extract contours and machining positions from DXF files (R12 format).	User's Manual for Conversational Programming

Global Program Settings software option	Description
Function for superimposing coordinate transformations in the Program Run modes, handwheel superimposed traverse in virtual axis direction.	User's Manual for Conversational Programming

AFC software option	Description
Function for adaptive feed-rate control for optimizing the machining conditions during series production.	User's Manual for Conversational Programming

KinematicsOpt software option	Description
Touch-probe cycles for inspecting and optimizing the machine accuracy	Page 480

3D-ToolComp software option	Description
3-D radius compensation depending on the tool's contact angle for **LN** blocks.	User's Manual for Conversational Programming

Expanded Tool Management software option	Description
Tool management that can be changed by the machine manufacturer using Python scripts	User's Manual for Conversational Programming

CAD Viewer software option	Description
Opening of 3-D models on the control	User's Manual for Conversational Programming

Interpolation turning software option	Description
Interpolation turning of a shoulder with Cycle 290.	Page 323

Remote Desktop Manager software option	Description
Remote operation of external computer units (e.g. a Windows PC) via the user interface of the TNC	User's Manual for Conversational Programming

Cross Talk Compensation (CTC) software option	Description
Compensation of axis couplings	Machine Manual

Position Adaptive Control (PAC) software option	Description
Adaptation of control parameters	Machine Manual

Load Adaptive Control (LAC) software option	Description
Dynamic changing of control parameters	Machine Manual

Active Chatter Control (ACC) software option	Description
Fully automatic function for chatter control during machining	Machine Manual

Feature content level (upgrade functions)

Along with software options, significant further improvements of the TNC software are managed via the Feature Content Level upgrade functions. Functions subject to the FCL are not available simply by updating the software on your TNC.

All upgrade functions are available to you without surcharge when you receive a new machine.

Upgrade functions are identified in the manual with **FCL n,** where **n** indicates the sequential number of the feature content level.

You can purchase a code number in order to permanently enable the FCL functions. For more information, contact your machine tool builder or HEIDENHAIN.

FCL 4 functions	Description
Graphical depiction of the protected space when DCM collision monitoring is active	User's Manual
Handwheel superimposition in stopped condition when DCM collision monitoring is active	User's Manual
3-D basic rotation (set-up compensation)	Machine manual

FCL 3 functions	Description
Touch probe cycle for 3-D probing	Page 469
Touch probe cycles for automatic datum setting using the center of a slot/ridge	Page 363
Feed-rate reduction for the machining of contour pockets with the tool being in full contact with the workpiece	User's Manual
PLANE function: Entry of axis angle	User's Manual
User documentation as a context-sensitive help system	User's Manual
smarT.NC: Programming of smarT.NC and machining can be carried out simultaneously	User's Manual
smarT.NC: Contour pocket on point pattern	smarT.NC Pilot

FCL 3 functions	Description
smarT.NC: Preview of contour programs in the file manager	smarT.NC Pilot
smarT.NC: Positioning strategy for machining point patterns	smarT.NC Pilot

FCL 2 functions	Description
3-D line graphics	User’s Manual
Virtual tool axis	User’s Manual
USB block devices supported (memory sticks, hard disks, CD-ROM drives)	User’s Manual
Filtering of externally created contours	User’s Manual
Possibility of assigning different depths to each subcontour in the contour formula	User’s Manual
DHCP dynamic IP address management	User’s Manual
Touch-probe cycle for global setting of touch-probe parameters	Page 474
smarT.NC: Graphic support of block scan	smarT.NC Pilot
smarT.NC: Coordinate transformation	smarT.NC Pilot
smarT.NC: PLANE function	smarT.NC Pilot

Intended place of operation

The TNC complies with the limits for a Class A device in accordance with the specifications in EN 55022, and is intended for use primarily in industrially-zoned areas.

New cycle functions of software 60642x-01

- New Cycle 275 "Trochoidal Contour Slot"(see "TROCHOIDAL SLOT (Cycle 275, DIN/ISO: G275)" on page 210)
- In Cycle 241 "Single-Lip Deep-Hole Drilling" it is now possible to define a dwell depth (see "SINGLE-LIP DEEP-HOLE DRILLING (Cycle 241, DIN/ISO: G241)" on page 98)
- The approach and departure behavior of Cycle 39 "Cylinder Surface Contour" can now be adjusted (see "Cycle run" on page 238)
- New touch probe cycle for calibration of a touch probe on a calibration sphere (see "CALIBRATE TS (Cycle 460, DIN/ISO: G460)" on page 476)
- KinematicsOpt: An additional parameter for determination of the backlash in a rotary axis was introduced (see "Backlash" on page 491)
- KinematicsOpt: Better support for positioning of Hirth-coupled axes (see "Machines with Hirth-coupled axes" on page 487)

New cycle functions of software 60642x-02

- New Cycle **225 Engraving** (see "ENGRAVING (Cycle 225, DIN/ISO: G225)" on page 319)
- New Cycle **276 3-D Contour Train** (see "THREE-D CONTOUR TRAIN (Cycle 276, DIN/ISO: G276)" on page 215)
- New Cycle **290 Interpolation turning** (see "INTERPOLATION TURNING (software option, Cycle 290, DIN/ISO: G290)" on page 323)
- In the thread milling cycles 26x a separate feed rate is now available for tangential approach to the thread (see the descriptions of the respective cycles)
- The following improvements were made to the KinematicsOpt cycles:
  - Newer, faster optimization algorithm
  - After angle optimization, a separate measurement series is no longer required for position optimization (see "Various modes (Q406)" on page 496)
  - Return of the offset errors (change of machine datum) to the parameters Q147-149 (see "Cycle run" on page 484)
  - Up to eight plane measuring points for ball measurement (see "Cycle parameters" on page 493)
  - Rotary axes that are not configured are ignored by the TNC when executing the cycle (see "Please note while programming:" on page 492)

New cycle functions of software 60642x-03

- With Cycle 256 Rectangular Stud, a parameter is now available with which you can determine the approach position on the stud (see "RECTANGULAR STUD (Cycle 256, DIN/ISO: G256)" on page 161)
- With Cycle 257 Circular Stud Milling, a parameter is now available with which you can determine the approach position on the stud (see "CIRCULAR STUD (Cycle 257, DIN/ISO: G257)" on page 165)

New cycle functions of software 60642x-04

- Cycle 25: Automatic identification of residual material added (see "CONTOUR TRAIN (Cycle 25, DIN/ISO: G125)" on page 206)
- Cycle 200: Input parameter Q359 added to allow definition of the depth reference (see "DRILLING (Cycle 200)" on page 75)
- Cycle 203: Input parameter Q359 added to allow definition of the depth reference (see "UNIVERSAL DRILLING (Cycle 203, DIN/ISO: G203)" on page 83)
- Cycle 205: Input parameter Q208 for retraction feed rate added (see "UNIVERSAL PECKING (Cycle 205, DIN/ISO: G205)" on page 91)
- Cycle 205: Input parameter Q359 added to allow definition of the depth reference (see "UNIVERSAL PECKING (Cycle 205, DIN/ISO: G205)" on page 91)
- Cycle 225: Input of German umlauts is now possible; text can also be arranged at an angle (see "ENGRAVING (Cycle 225, DIN/ISO: G225)" on page 319)
- Cycle 253: Input parameter Q439 for feed rate reference added (see "SLOT MILLING (Cycle 253, DIN/ISO: G253)" on page 150)
- Cycle 254: Input parameter Q439 for feed rate reference added (see "CIRCULAR SLOT (Cycle 254, DIN/ISO: G254)" on page 155)
- Cycle 276: Automatic identification of residual material added (see "THREE-D CONTOUR TRAIN (Cycle 276, DIN/ISO: G276)" on page 215)
- Cycle 290: It is now also possible to machine a recess with Cycle 290 (see "INTERPOLATION TURNING (software option, Cycle 290, DIN/ISO: G290)" on page 323)
- Cycle 404: Input parameter Q305 added to allow saving a basic rotation in any line of the datum table (see "SET BASIC ROTATION (Cycle 404, DIN/ISO: G404)" on page 353)

New cycle functions of software 60642x-04 SP8

- With Cycle 253, Slot Milling, a parameter is now available with which you can define the feed rate reference when machining the slot (see "SLOT MILLING (Cycle 253, DIN/ISO: G253)" on page 150)
- With Cycle 254, Circular Slot, a parameter is now available with which you can define the feed rate reference when machining the slot (see "CIRCULAR SLOT (Cycle 254, DIN/ISO: G254)" on page 155)

Changed cycle functions of software 60642x-01

- The approach behavior during side finishing with Cycle 24 (DIN/ISO: G124) was changed (see "Please note while programming:" on page 202)

Changed cycle functions of software 60642x-02

- Position of the soft key for defining Cycle 270 has been changed

Changed cycle functions of software 60642x-04

- Cycle 206: The TNC now monitors the thread pitch if entered in the tool table
- Cycle 207: The TNC now monitors the thread pitch if entered in the tool table
- Cycle 209: The TNC now monitors the thread pitch if entered in the tool table
- Cycle 209: The TNC now completely retracts the tool from the hole during chip breaking if parameter Q256 is defined to be 0 (retraction distance for chip breaking)
- Cycle 202: The TNC now does not retract the tool from the hole bottom if parameter Q214 is defined to be 0 (disengaging direction)
- Cycle 405: The TNC now also writes the datum in line 0 of the datum table if the parameter Q337=0 is defined
- Corresponding touch-probe cycles 4xx: The input range of parameter Q305 (Reference point number or Datum number) was extended to 99999
- Cycle 451 and Cycle 452: During a measurement, the TNC now closes the status window only if the distance to the calibration sphere is greater than the radius of the ball tip

Contents

4 Fixed Cycles: Tapping / Thread Milling 105

6 Fixed Cycles: Pattern Definitions 173

1

Fundamentals / Overviews

1.1 Introduction

Frequently recurring machining cycles that comprise several working steps are stored in the TNC memory as standard cycles. Coordinate transformations and several special functions are also available as cycles.

Most cycles use Q parameters as transfer parameters. Parameters with specific functions that are required in several cycles always have the same number: For example, **Q200** is always assigned the set-up clearance, **Q202** the plunging depth, etc.

Danger of collision!

Cycles sometimes execute extensive operations. For safety reasons, you should run a graphical program test before machining.

If you use indirect parameter assignments in cycles with numbers greater than 200 (e.g. **Q210 = Q1**), any change in the assigned parameter (e.g. Q1) will have no effect after the cycle definition. Define the cycle parameter (e.g. **Q210**) directly in such cases.

If you define a feed-rate parameter for fixed cycles greater than 200, then instead of entering a numerical value you can use soft keys to assign the feed rate defined in the **TOOL CALL** block (FAUTO soft key). You can also use the feed-rate alternatives **FMAX** (rapid traverse), **FZ** (feed per tooth) and **FU** (feed per rev), depending on the respective cycle and the function of the feed-rate parameter.

Note that, after a cycle definition, a change of the **FAUTO** feed rate has no effect, because internally the TNC assigns the feed rate from the **TOOL CALL** block when processing the cycle definition.

If you want to delete a block that is part of a cycle, the TNC asks you whether you want to delete the whole cycle.

1.2 Available cycle groups

Overview of fixed cycles

CYCL DEF

▶ The soft key row shows the available groups of cycles

Cycle group	Soft key	Page
Cycles for pecking, reaming, boring, and counterboring	DRILLING/ THREAD	Page 72
Cycles for tapping, thread cutting and thread milling	DRILLING/ THREAD	Page 106
Cycles for milling pockets, studs and slots	POCKETS/ STUDS/ SLOTS	Page 140
Cycles for producing hole patterns, such as circular or linear point patterns	PATTERN	Page 174
SL (Subcontour List) cycles which allow the contour-parallel machining of relatively complex contours consisting of several overlapping subcontours, cylinder surface interpolation	SL II	Page 186
Cycles for multipass milling of flat or twisted surfaces	MULTIPASS MILLING	Page 262
Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours	COORD. TRANSF.	Page 280
Special cycles such as dwell time, program call, oriented spindle stop, tolerance, engraving and interpolation turning (option)	SPECIAL CYCLES	Page 310

▷

▶ If required, switch to machine-specific fixed cycles. These fixed cycles can be integrated by your machine tool builder.

Overview of touch probe cycles

TOUCH PROBE

▶ The soft key row shows the available groups of cycles

Cycle group	Soft key	Page
Cycles for automatic measurement and compensation of workpiece misalignment		Page 338
Cycles for automatic workpiece presetting		Page 360
Cycles for automatic workpiece inspection		Page 414
Calibration cycles, special cycles	SPECIAL CYCLES	Page 464
Cycles for automatic kinematics measurement	KINEMATICS	Page 480
Cycles for automatic tool measurement (enabled by the machine tool builder)		Page 512

▶ If required, switch to machine-specific touch probe cycles. These touch probe cycles can be integrated by your machine tool builder.

2

Using Fixed Cycles

2.1 Working with fixed cycles

General information

If you transfer NC programs from old TNC controls or create NC programs externally (by using a CAM system or an ASCI editor, for example), keep the following conventions in mind:

- Fixed cycles and touch probe cycles with numbers **smaller** than 200:
  - In older iTNC software versions and older TNC controls, text strings that could not always be converted correctly by the current iTNC editor were used in some conversational languages. Make sure that cycle texts do not end with a period.
- Fixed cycles and touch probe cycles with numbers **greater** than 200:
  - Indicate the end of a line with the tilde character (**~**). The last parameter in the cycle must not contain any tilde character.
  - Cycle names and cycle comments do not essentially need to be indicated. The iTNC supplements the cycle names and cycle comments in the selected conversational language when the program is transferred to the control.

Machine-specific cycles

In addition to the HEIDENHAIN cycles, many machine tool builders offer their own cycles in the TNC. These cycles are available in a separate cycle-number range:

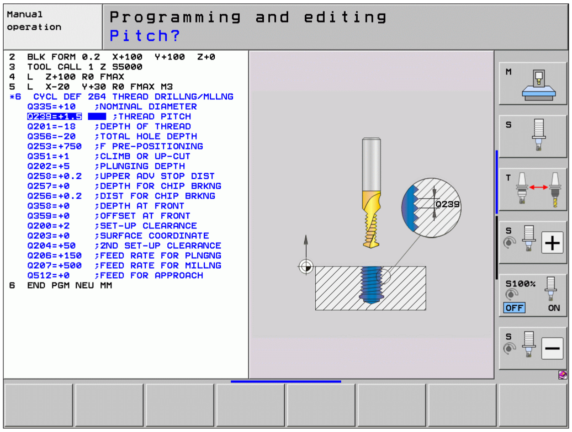
- Cycles 300 to 399
  Machine-specific cycles that are to be defined through the CYCLE DEF key
- Cycles 500 to 599
  Machine-specific touch probe cycles that are to be defined through the TOUCH PROBE key

Refer to your machine manual for a description of the specific function.

Sometimes machine-specific cycles use transfer parameters that HEIDENHAIN already uses in standard cycles. The TNC executes DEF-active cycles as soon as they are defined (see also "Calling cycles" on page 51). It executes CALL-active cycles only after they have been called (see also "Calling cycles" on page 51). When DEF-active cycles and CALL-active cycles are used simultaneously, it is important to prevent overwriting of transfer parameters already in use. Use the following procedure:

- As a rule, always program DEF-active cycles before CALL-active cycles
- If you do want to program a DEF-active cycle between the definition and call of a CALL-active cycle, do it only if there is no common use of specific transfer parameters

Defining a cycle using soft keys



CYCL DEF

- The soft-key row shows the available groups of cycles

DRILLING/ THREAD

- Press the soft key for the desired group of cycles, for example DRILLING for the drilling cycles

262

- Select the desired cycle, for example THREAD MILLING. The TNC initiates the programming dialog and asks for all required input values. At the same time a graphic of the input parameters is displayed in the right screen window. The parameter that is asked for in the dialog prompt is highlighted.
- Enter all parameters requested by the TNC and conclude each entry with the ENT key
- The TNC ends the dialog when all required data has been entered

Defining a cycle using the GOTO function

CYCL DEF

- The soft-key row shows the available groups of cycles

GOTO

- The TNC shows an overview of cycles in a pop-up window
- Choose the desired cycle with the arrow keys, or
- Choose the desired cycle with CTRL and the arrow keys (for pagewise scrolling), or
- Enter the cycle number and confirm it with the ENT key. The TNC then initiates the cycle dialog as described above

Example NC blocks

```
7 CYCL DEF 200 DRILLING
    Q200=2    ;SET-UP CLEARANCE
    Q201=3    ;DEPTH
    Q206=150  ;FEED RATE FOR PLNGNG
    Q202=5    ;PLUNGING DEPTH
    Q210=0    ;DWELL TIME AT TOP
    Q203=+0   ;SURFACE COORDINATE
    Q204=50   ;2ND SET-UP CLEARANCE
    Q211=0.25 ;DWELL TIME AT DEPTH
```

Calling cycles

Requirements

The following data must always be programmed before a cycle call:

- **BLK FORM** for graphic display (needed only for test graphics)
- Tool call
- Direction of spindle rotation (M functions M3/M4)
- Cycle definition (CYCL DEF)

For some cycles, additional prerequisites must be observed. They are detailed in the descriptions for each cycle.

The following cycles become effective automatically as soon as they are defined in the part program. These cycles cannot and must not be called:

- Cycle 220 for point patterns on circles and Cycle 221 for point patterns on lines
- SL Cycle 14 CONTOUR
- SL Cycle 20 CONTOUR DATA
- Cycle 32 TOLERANCE
- Coordinate transformation cycles
- Cycle 9 DWELL TIME
- All touch probe cycles

You can call all other cycles with the functions described as follows.

Calling a cycle with CYCL CALL

The **CYCL CALL** function calls the most recently defined fixed cycle once. The starting point of the cycle is the position that was programmed last before the CYCL CALL block.

CYCL CALL

- ▶ Program the cycle call: Press the CYCL CALL key
- ▶ Enter the cycle call: Press the CYCL CALL M soft key
- ▶ If necessary, enter the miscellaneous function M (for example **M3** to switch the spindle on), or end the dialog by pressing the END key

Calling a cycle with CYCL CALL PAT

The **CYCL CALL PAT** function calls the most recently defined fixed cycle at all positions that you defined in a PATTERN DEF pattern definition (see "PATTERN DEF pattern definition" on page 59) or in a point table (see "Point tables" on page 67).

Calling a cycle with CYCL CALL POS

The **CYCL CALL POS** function calls the most recently defined fixed cycle once. The starting point of the cycle is the position that you defined in the **CYCL CALL POS** block.

Using positioning logic the TNC moves to the position defined in the **CYCL CALL POS** block:

- If the tool's current position in the tool axis is greater than the top surface of the workpiece (Q203), the TNC moves the tool to the programmed position first in the working plane and then in the tool axis.
- If the tool's current position in the tool axis is below the top surface of the workpiece (Q203), the TNC moves the tool to the programmed position first in the tool axis to the clearance height and then in the working plane to the programmed position.

Three coordinate axes must always be programmed in the **CYCL CALL POS** block. With the coordinate in the tool axis you can easily change the starting position. It serves as an additional datum shift.

The feed rate most recently defined in the **CYCL CALL POS** block applies only for traverse to the start position programmed in this block.

As a rule, the TNC moves without radius compensation (R0) to the position defined in the **CYCL CALL POS** block.

If you use CYCL CALL POS to call a cycle in which a start position is defined (for example Cycle 212), then the position defined in the cycle serves as an additional shift of the position defined in the CYCL CALL POS block. You should therefore always define the start position to be set in the cycle as 0.

Calling a cycle with M99/M89

The **M99** function, which is active only in the block in which it is programmed, calls the last defined fixed cycle once. You can program **M99** at the end of a positioning block. The TNC moves to this position and then calls the last defined fixed cycle.

If the TNC is to execute the cycle automatically after every positioning block, program the first cycle call with **M89** (depending on MP7440).

To cancel the effect of M89, program:

- **M99** in the positioning block in which you move to the last starting point, or
- A **CYCL CALL POS** block or
- A new fixed cycle with **CYCL DEF**

Working with the secondary axes U/V/W

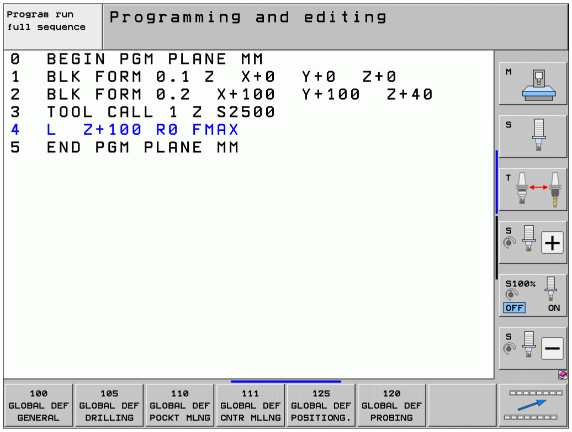
The TNC performs infeed movements in the axis that was defined in the TOOL CALL block as the spindle axis. It performs movements in the working plane only in the principal axes X, Y or Z. Exceptions:

- You program secondary axes for the side lengths in Cycles 3 SLOT MILLING and 4 POCKET MILLING.
- You program secondary axes in the first block of the contour geometry subprogram of an SL cycle.
- In Cycles 5 (CIRCULAR POCKET), 251 (RECTANGULAR POCKET), 252 (CIRCULAR POCKET), 253 (SLOT) and 254 (CIRCULAR SLOT), the TNC machines the cycle in the axes that you programmed in the last positioning block before the cycle call. When tool axis Z is active, the following combinations are permissible:
  - X/Y
  - X/V
  - U/Y
  - U/V

2.2 Program defaults for cycles

Overview

All Cycles 20 to 25, as well as all of those with numbers 200 or higher, always use identical cycle parameters, such as the set-up clearance **Q200,** which you must enter for each cycle definition. The **GLOBAL DEF** function gives you the possibility of defining these cycle parameters once at the beginning of the program, so that they are effective globally for all fixed cycles used in the program. In the respective fixed cycle you then simply link to the value defined at the beginning of the program.



The following GLOBAL DEF functions are available:

Machining patterns	Soft key	Page
GLOBAL DEF COMMON Definition of generally valid cycle parameters	100 GLOBAL DEF GENERAL	Page 56
GLOBAL DEF DRILLING Definition of specific drilling cycle parameters	105 GLOBAL DEF DRILLING	Page 56
GLOBAL DEF POCKET MILLING Definition of specific pocket-milling cycle parameters	110 GLOBAL DEF POCKT MLNG	Page 57
GLOBAL DEF CONTOUR MILLING Definition of specific contour milling parameters	111 GLOBAL DEF CNTR MLLNG	Page 57
GLOBAL DEF POSITIONING Definition of the positioning behavior for **CYCL CALL PAT**	125 GLOBAL DEF POSITIONG.	Page 57
GLOBAL DEF PROBING Definition of specific touch probe cycle parameters	120 GLOBAL DEF PROBING	Page 58

Use the INSERT SMART UNIT function (see Special Functions chapter in the Conversational Programming User's Manual) and then **UNIT 700** to insert all GLOBAL DEF functions into a block.

Entering GLOBAL DEF definitions

- Select the Programming and Editing operating mode

SPEC FCT

- Press the special functions key

PROGRAM DEFAULTS

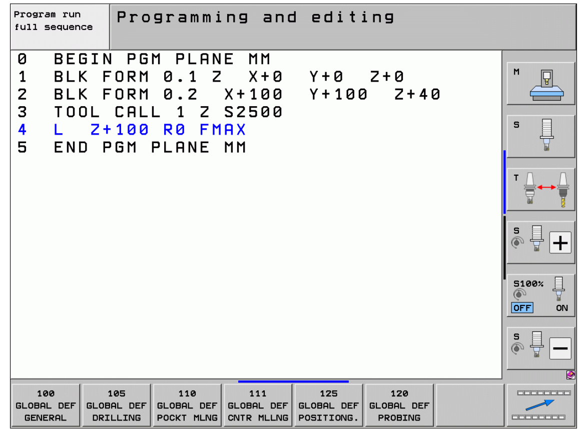
- Select the functions for program defaults

GLOBAL DEF

- Select **GLOBAL DEF** functions

100 GLOBAL DEF GENERAL

- Select the desired GLOBAL DEF function, e.g. **GLOBAL DEF COMMON**
- Enter the required definitions, and confirm each entry with the ENT key



Using GLOBAL DEF information

If you have entered the corresponding GLOBAL DEF functions at the beginning of the program, then you can link to these globally valid values when defining any fixed cycle.

Proceed as follows:

- Select the Programming and Editing operating mode

CYCL DEF

- Select fixed cycles

DRILLING/ THREAD

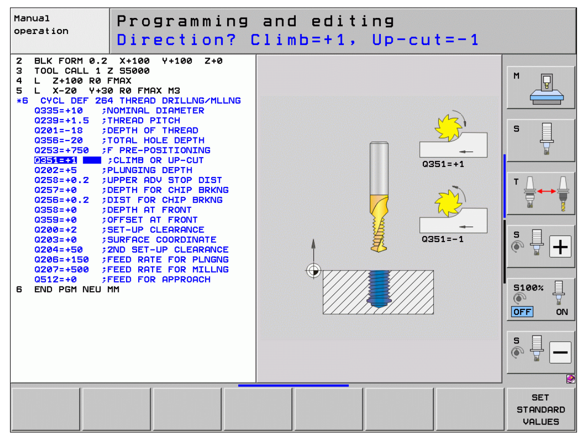
- Select the desired group of cycles, for example: drilling cycles

200

- Select the desired cycle, e.g. **DRILLING**
- The TNC displays the SET STANDARD VALUES soft key, if there is a global parameter for it

SET STANDARD VALUES

- Press the SET STANDARD VALUES soft key. The TNC enters the word **PREDEF** (predefined) in the cycle definition. You have now created a link to the corresponding **GLOBAL DEF** parameter that you defined at the beginning of the program



> **Danger of collision!**
>
> Please note that later changes to the program settings affect the entire machining program, and can therefore change the machining procedure significantly.
>
> If you enter a fixed value in a fixed cycle, then this value will not be changed by the **GLOBAL DEF** functions.

Global data valid everywhere

- **Set-up clearance**: Distance between tool tip and workpiece surface for automated approach of the cycle start position in the tool axis
- **2nd set-up clearance**: Position to which the TNC positions the tool at the end of a machining step. The next machining position is approached at this height in the machining plane
- **F positioning**: Feed rate at which the TNC traverses the tool within a cycle
- **F retraction**: Feed rate at which the TNC retracts the tool

The parameters are valid for all fixed cycles with numbers greater than 2xx.

Global data for drilling operations

- **Retraction rate for chip breaking**: Value by which the TNC retracts the tool during chip breaking
- **Dwell time at depth**: Time in seconds that the tool remains at the hole bottom
- **Dwell time at top**: Time in seconds that the tool remains at the set-up clearance

The parameters apply to the drilling, tapping and thread milling cycles 200 to 209, 240, and 262 to 267.

Global data for milling operations with pocket cycles 25x

- **Overlap factor**: The tool radius multiplied by the overlap factor equals the lateral stepover
- **Climb or up-cut**: Select the type of milling
- **Plunging type**: Plunge into the material helically, in a reciprocating motion, or vertically

> The parameters apply to milling cycles 251 to 257.

Global data for milling operations with contour cycles

- **Set-up clearance**: Distance between tool tip and workpiece surface for automated approach of the cycle start position in the tool axis
- **Clearance height**: Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle)
- **Overlap factor**: The tool radius multiplied by the overlap factor equals the lateral stepover
- **Climb or up-cut**: Select the type of milling

> The parameters apply to SL cycles 20, 22, 23, 24 and 25.

Global data for positioning behavior

- **Positioning behavior**: Retraction in the tool axis at the end of the machining step: Return to the 2nd set-up clearance or to the position at the beginning of the unit

> The parameters apply to each fixed cycle that you call with the **CYCL CALL PAT** function.

Global data for probing functions

- **Set-up clearance**: Distance between stylus and workpiece surface for automated approach of the probing position
- **Clearance height**: The coordinate in the touch probe axis to which the TNC traverses the touch probe between measuring points, if the **Move to clearance height** option is activated
- **Move to clearance height**: Select whether the TNC moves the touch probe to the set-up clearance or clearance height between the measuring points

> The parameters apply to all touch probe cycles with numbers greater than 4xx.

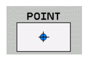
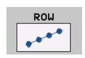
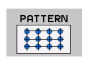
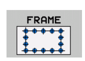
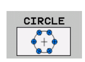
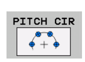
2.3 PATTERN DEF pattern definition

Application

You use the **PATTERN DEF** function to easily define regular machining patterns, which you can call with the **CYCL CALL PAT** function. As with the cycle definitions, support graphics that illustrate the respective input parameter are also available for pattern definitions.

> **PATTERN DEF** is to be used only in connection with the tool axis Z.

The following machining patterns are available:

Machining pattern	Soft key	Page
POINT Definition of up to any 9 machining positions		Page 61
ROW Definition of a single row, straight or rotated		Page 62
PATTERN Definition of a single pattern, straight, rotated or distorted		Page 63
FRAME Definition of a single frame, straight, rotated or distorted		Page 64
CIRCLE Definition of a full circle		Page 65
PITCH CIRCLE Definition of a pitch circle		Page 66

Entering PATTERN DEF

- Select the Programming and Editing operating mode
- Press the special functions key
- Select the functions for contour and point machining
- Open a **PATTERN DEF** block
- Select the desired machining pattern, e.g. a single row
- Enter the required definitions, and confirm each entry with the ENT key

Using PATTERN DEF

As soon as you have entered a pattern definition, you can call it with the **CYCL CALL PAT** function (see "Calling a cycle with CYCL CALL PAT" on page 51). The TNC then performs the most recently defined machining cycle on the machining pattern you defined.

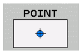
A machining pattern remains active until you define a new one, or select a point table with the **SEL PATTERN** function.

You can use the mid-program startup function to select any point at which you want to start or continue machining (see User's Manual, Test Run and Program Run sections).

Defining individual machining positions

You can enter up to 9 machining positions. Confirm each entry with the ENT key.

If you have defined a **workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

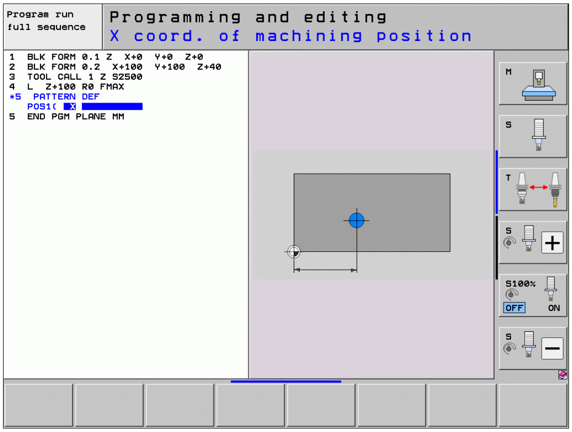


- **X coord. of machining position** (absolute): Enter X coordinate
- **Y coord. of machining position** (absolute): Enter Y coordinate
- **Workpiece surface coordinate** (absolute): Enter Z coordinate at which machining is to begin

Example: NC blocks

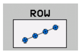
```
10 L Z+100 R0 FMAX
```

```
11 PATTERN DEF
POS1 (X+25 Y+33.5 Z+0)
POS2 (X+50 Y+75 Z+0)
```



Defining a single row

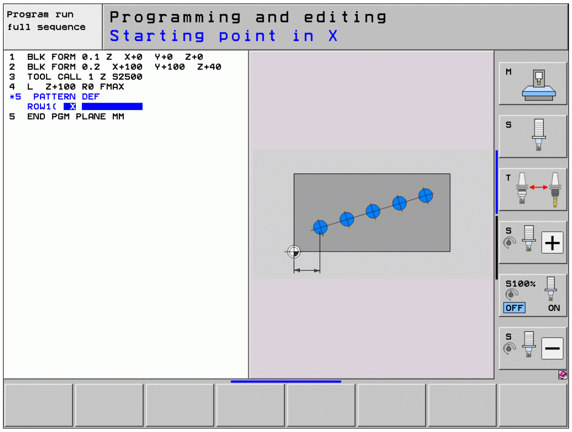
If you have defined a **workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



- **Starting point in X** (absolute): Coordinate of the starting point of the row in the X axis
- **Starting point in Y** (absolute): Coordinate of the starting point of the row in the Y axis
- **Spacing of machining positions (incremental)**: Distance between the machining positions. You can enter a positive or negative value
- **Number of repetitions**: Total number of machining operations
- **Rot. position of entire pattern (absolute)**: Angle of rotation around the entered starting point. Reference axis: Reference axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value
- **Workpiece surface coordinate** (absolute): Enter Z coordinate at which machining is to begin

Example: NC blocks

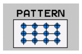
```
10 L Z+100 R0 FMAX
11 PATTERN DEF
ROW1 (X+25 Y+33.5 D+8 NUM5 ROT+0 Z+0)
```



Defining a single pattern

If you have defined a **workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **rotated position** of the entire pattern.

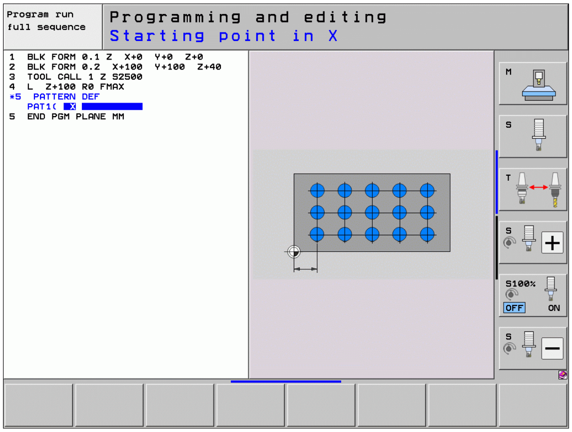


- **Starting point in X** (absolute): Coordinate of the starting point of the pattern in the X axis
- **Starting point in Y** (absolute): Coordinate of the starting point of the pattern in the Y axis
- **Spacing of machining positions X (incremental)**: Distance between the machining positions in the X direction. You can enter a positive or negative value
- **Spacing of machining positions Y (incremental)**: Distance between the machining positions in the Y direction. You can enter a positive or negative value
- **Number of columns**: Total number of columns in the pattern
- **Number of lines**: Total number of rows in the pattern
- **Rot. position of entire pattern (absolute)**: Angle of rotation by which the entire pattern is rotated around the entered starting point. Reference axis: Reference axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value
- **Rotary pos. ref. ax.**: Angle of rotation around which only the reference axis of the machining plane is distorted with respect to the entered starting point. You can enter a positive or negative value.
- **Rotary pos. minor ax.**: Angle of rotation around which only the minor axis of the machining plane is distorted with respect to the entered starting point. You can enter a positive or negative value.
- **Workpiece surface coordinate** (absolute): Enter Z coordinate at which machining is to begin

Example: NC blocks

```
10 L Z+100 R0 FMAX
```

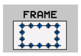
```
11 PATTERN DEF
PAT1 (X+25 Y+33.5 DX+8 DY+10 NUMX5
NUMY4 ROT+0 ROTX+0 ROTY+0 Z+0)
```



Defining individual frames

If you have defined a **`workpiece surface in Z`** not equal to 0, then this value is effective in addition to the workpiece surface **`Q203`** that you defined in the machining cycle.

The **`Rotary pos. ref. ax.`** and **`Rotary pos. minor ax.`** parameters are added to a previously performed **`rotated position`** of the entire pattern.

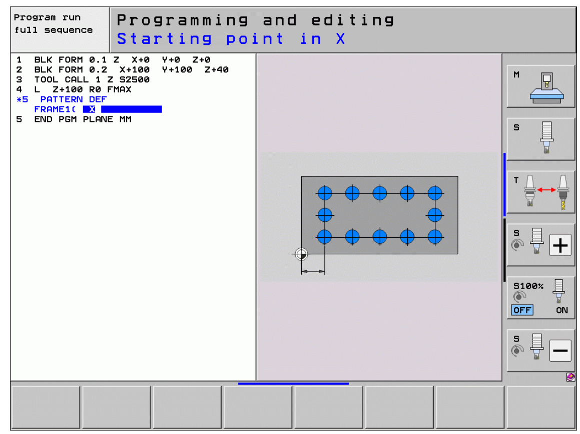


- **`Starting point in X`** (absolute): Coordinate of the starting point of the frame in the X axis
- **`Starting point in Y`** (absolute): Coordinate of the starting point of the frame in the Y axis
- **`Spacing of machining positions X (incremental)`**: Distance between the machining positions in the X direction. You can enter a positive or negative value
- **`Spacing of machining positions Y (incremental)`**: Distance between the machining positions in the Y direction. You can enter a positive or negative value
- **`Number of columns`**: Total number of columns in the pattern
- **`Number of lines`**: Total number of rows in the pattern
- **`Rot. position of entire pattern (absolute)`**: Angle of rotation by which the entire pattern is rotated around the entered starting point. Reference axis: Reference axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value
- **`Rotary pos. ref. ax.`**: Angle of rotation around which only the reference axis of the machining plane is distorted with respect to the entered starting point. You can enter a positive or negative value.
- **`Rotary pos. minor ax.`**: Angle of rotation around which only the minor axis of the machining plane is distorted with respect to the entered starting point. You can enter a positive or negative value.
- **`Workpiece surface coordinate`** (absolute): Enter Z coordinate at which machining is to begin

Example: NC blocks

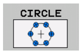
```
10 L Z+100 R0 FMAX
```

```
11 PATTERN DEF
FRAME1 (X+25 Y+33.5 DX+8 DY+10 NUMX5
NUMY4 ROT+0 ROTX+0 ROTY+0 Z+0)
```



Defining a full circle

If you have defined a **workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

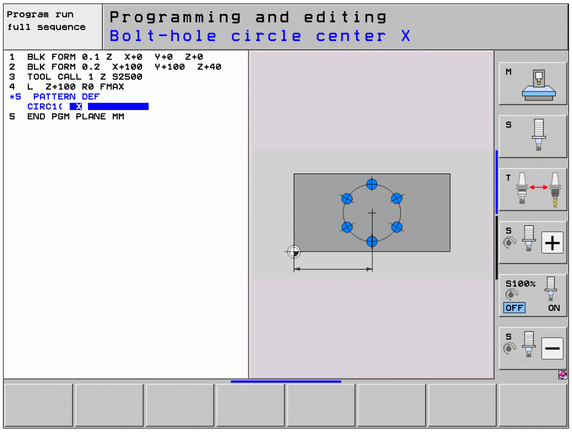


- **Bolt-hole circle center X** (absolute): Coordinate of the circle center in the X axis
- **Bolt-hole circle center Y** (absolute): Coordinate of the circle center in the Y axis
- **Bolt-hole circle diameter**: Diameter of the bolt-hole circle
- **Starting angle**: Polar angle of the first machining position. Reference axis: Reference axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value
- **Number of repetitions**: Total number of machining positions on the circle
- **Workpiece surface coordinate** (absolute): Enter Z coordinate at which machining is to begin

Example: NC blocks

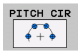
```
10 L Z+100 R0 FMAX
```

```
11 PATTERN DEF
CIRC1 (X+25 Y+33 D80 START+45 NUM8 Z+0)
```



Defining a pitch circle

If you have defined a **workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

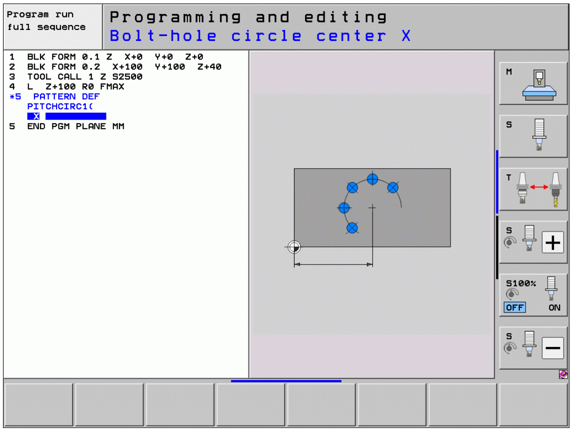


- **Bolt-hole circle center X** (absolute): Coordinate of the circle center in the X axis
- **Bolt-hole circle center Y** (absolute): Coordinate of the circle center in the Y axis
- **Bolt-hole circle diameter**: Diameter of the bolt-hole circle
- **Starting angle**: Polar angle of the first machining position. Reference axis: Reference axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value
- **Stepping angle/end angle**: Incremental polar angle between two machining positions. You can enter a positive or negative value. As an alternative you can enter the end angle (switch via soft key).
- **Number of repetitions**: Total number of machining positions on the circle
- **Workpiece surface coordinate** (absolute): Enter Z coordinate at which machining is to begin

Example: NC blocks

```
10 L Z+100 R0 FMAX
```

```
11 PATTERN DEF
PITCHCIRC1 (X+25 Y+33 D80 START+45 STEP30
NUM8 Z+0)
```



2.4 Point tables

Application

You should create a point table whenever you want to run a cycle, or several cycles in sequence, on an irregular point pattern.

If you are using drilling cycles, the coordinates of the working plane in the point table represent the hole centers. If you are using milling cycles, the coordinates of the working plane in the point table represent the starting-point coordinates of the respective cycle (e.g. center-point coordinates of a circular pocket). Coordinates in the spindle axis correspond to the coordinate of the workpiece surface.

Creating a point table

Select the **Programming and Editing** mode of operation.

PGM MGT — Call the file manager: Press the PGM MGT key

FILE NAME?

ENT — Enter the name and file type of the point table and confirm your entry with the ENT key.

MM — Select the unit of measure: Press the MM or INCH soft key. The TNC switches to the program blocks window and displays an empty point table

INSERT LINE — With the soft key INSERT LINE, insert new lines and enter the coordinates of the desired machining position

Repeat the process until all desired coordinates have been entered

Use the soft keys X OFF/ON, Y OFF/ON, Z OFF/ON (second soft-key row) to specify which coordinates you want to enter in the point table.

Hiding single points from the machining process

In the **FADE** column of the point table you can specify if the defined point is to be hidden during the machining process.

In the table, select the point to be hidden

Select the FADE column.

ENT Activate hiding, or

NO ENT Deactivate hiding

To hide the marked point during machining, you must also set the **Block skip** soft key to ON in the **Program Run** operating mode.

Defining the clearance height

In the **CLEARANCE** column you can define a separate height for every point. The TNC then positions the tool to this value in the tool axis before you reach this position in the working plane (see also "Calling a cycle in connection with point tables" on page 70).

Selecting a point table in the program

In the Programming and Editing mode of operation, select the program for which you want to activate the point table:

PGM CALL — Press the PGM CALL key to call the function for selecting the point table

POINT TABLE — Press the POINT TABLE soft key

SELECTION WINDOW — Press the WINDOW SELECTION soft key: The TNC superimposes a window where you can select the desired datum table

Select a point table with the arrow keys or by mouse click and confirm by pressing ENT: The TNC enters the complete path name in the **SEL PATTERN** block.

END — Conclude this function with the END key

Alternatively you can also enter the table name or the complete path name of the table to be called directly via the keyboard.

Example NC block

```
7 SEL PATTERN "TNC:\DIRKT5\NUST35.PNT"
```

Calling a cycle in connection with point tables

With **CYCL CALL PAT** the TNC runs the point table that you last defined (even if you defined the point table in a program that was nested with **CALL PGM.**

If you want the TNC to call the last defined fixed cycle at the points defined in a point table, then program the cycle call with **CYCL CALL PAT**:

CYCL CALL

- Program the cycle call: Press the CYCL CALL key
- Call the point table: Press the CYCL CALL PAT soft key
- Enter the feed rate at which the TNC is to move from point to point (if you make no entry the TNC will move at the last programmed feed rate; **FMAX** is not valid)
- If required, enter a miscellaneous function M, then confirm with the END key

The TNC retracts the tool to the clearance height between the starting points. The TNC uses either the spindle axis coordinate from the cycle call, the value from the cycle parameter Q204, or the value defined in the CLEARANCE column as the clearance height, whichever is greatest.

If you want to move at reduced feed rate when pre-positioning in the spindle axis, use the miscellaneous function M103.

Effect of the point tables with SL cycles and Cycle 12

The TNC interprets the points as an additional datum shift.

Effect of the point tables with Cycles 200 to 208 and 262 to 267

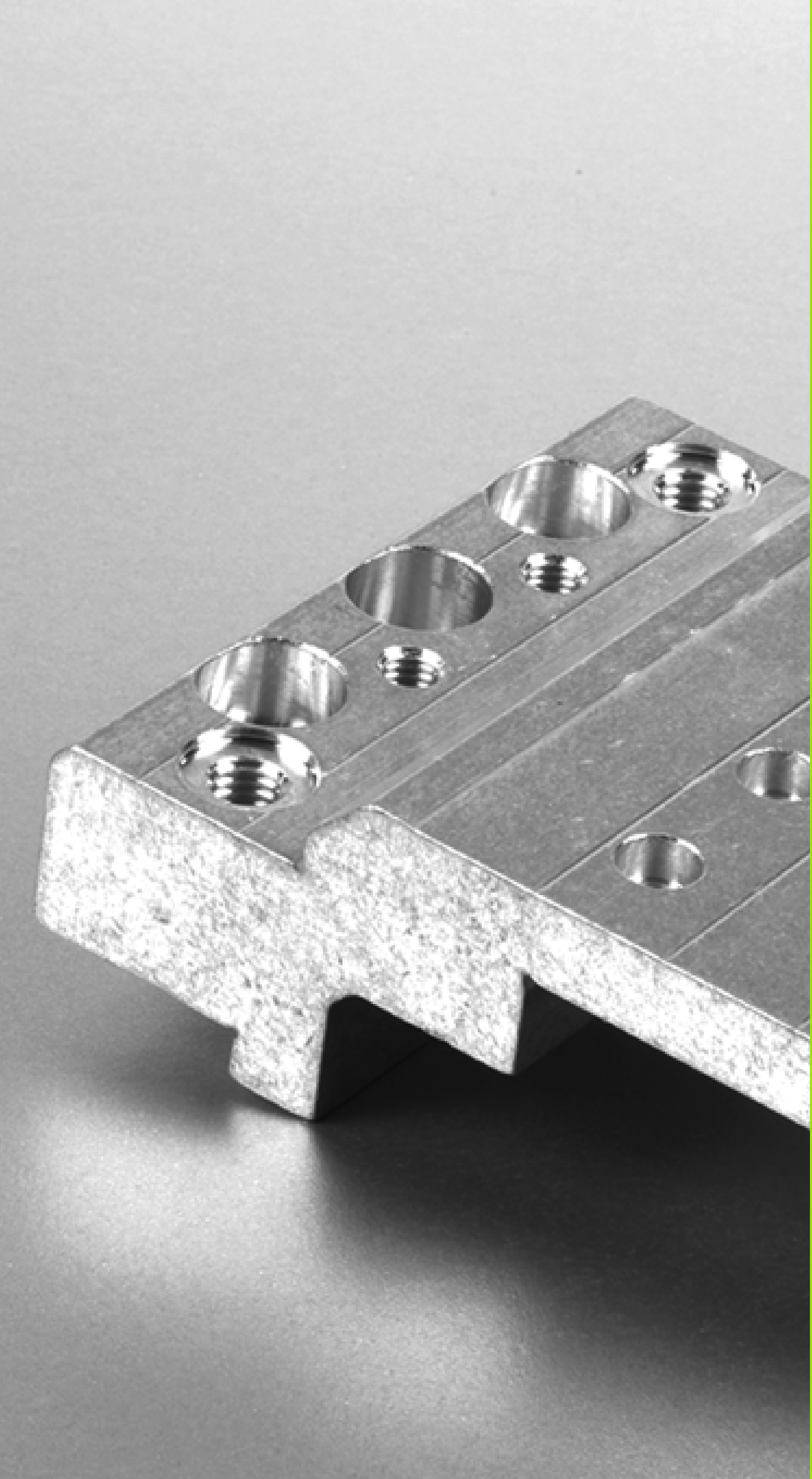
The TNC interprets the points of the working plane as coordinates of the hole centers. If you want to use the coordinate defined in the point table for the spindle axis as the starting point coordinate, you must define the workpiece surface coordinate (Q203) as 0.

Effect of the point tables with Cycles 210 to 215

The TNC interprets the points as an additional datum shift. If you want to use the points defined in the point table as starting-point coordinates, you must define the starting points and the workpiece surface coordinate (Q203) in the respective milling cycle as 0.

Effect of the point tables with Cycles 251 to 254

The TNC interprets the points of the working plane as coordinates of the cycle starting point. If you want to use the coordinate defined in the point table for the spindle axis as the starting point coordinate, you must define the workpiece surface coordinate (Q203) as 0.



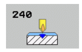
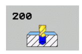
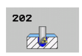
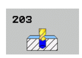
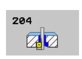
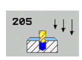
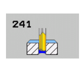
3

Fixed Cycles: Drilling

3.1 Fundamentals

Overview

The TNC offers 9 cycles for all types of drilling operations:

Cycle	Soft key	Page
240 CENTERING With automatic pre-positioning, 2nd set-up clearance, optional entry of the centering diameter or centering depth		Page 73
200 DRILLING With automatic pre-positioning, 2nd set-up clearance		Page 75
201 REAMING With automatic pre-positioning, 2nd set-up clearance		Page 77
202 BORING With automatic pre-positioning, 2nd set-up clearance		Page 79
203 UNIVERSAL DRILLING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and decrementing		Page 83
204 BACK BORING With automatic pre-positioning, 2nd set-up clearance		Page 87
205 UNIVERSAL PECKING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and advanced stop distance		Page 91
208 BORE MILLING With automatic pre-positioning, 2nd set-up clearance		Page 95
241 SINGLE-LIP DEEP-HOLE DRILLING With automatic pre-positioning to deepened starting point, shaft speed and coolant definition		Page 98

3.2 CENTERING (Cycle 240, DIN/ISO: G240)

Cycle run

1. The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the set-up clearance above the workpiece surface.
2. The tool is centered at the programmed feed rate **F** to the entered centering diameter or centering depth.
3. If defined, the tool remains at the centering depth.
4. Finally, the tool moves to set-up clearance or—if programmed—to the 2nd set-up clearance at rapid traverse **FMAX**.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter **Q344** (diameter) or **Q201** (depth) determines the working direction. If you program the diameter or depth = 0, the cycle will not be executed.

Danger of collision!

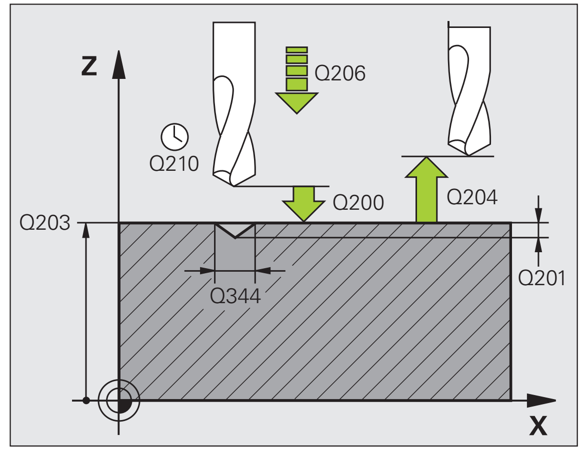
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

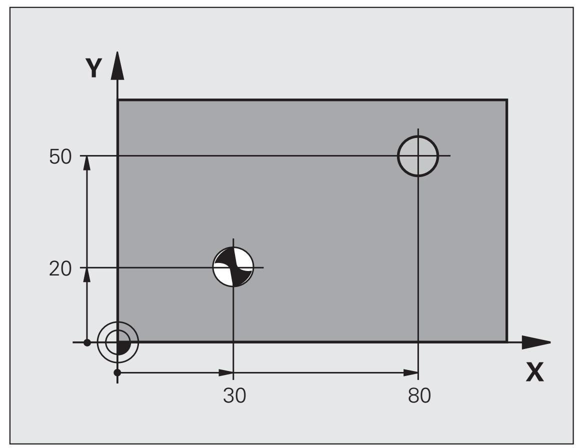
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive diameter or depth is entered.** This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

Cycle parameters

- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Enter a positive value. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Select Depth/Diameter (1/0)** Q343: Select whether centering is based on the entered diameter or depth. If the TNC is to center based on the entered diameter, the point angle of the tool must be defined in the **T-ANGLE** column of the tool table TOOL.T.
  0: Centering based on the entered depth
  1: Centering based on the entered diameter
- **Depth** Q201 (incremental): Distance between workpiece surface and centering bottom (tip of centering taper). Only effective if Q343=0 is defined. Input range -99999.9999 to 99999.9999
- **Diameter (algebraic sign)** Q344: Centering diameter. Only effective if Q343=1 is defined. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min during centering. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000; alternatively **PREDEF**
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**





Example: NC blocks

```
10 L Z+100 R0 FMAX
11 CYCL DEF 240 CENTERING
     Q200=2     ;SET-UP CLEARANCE
     Q343=1     ;SELECT DEPTH/DIA.
     Q201=+0    ;DEPTH
     Q344=-9    ;DIAMETER
     Q206=250   ;FEED RATE FOR PLNGNG
     Q211=0.1   ;DWELL TIME AT DEPTH
     Q203=+20   ;SURFACE COORDINATE
     Q204=100   ;2ND SET-UP CLEARANCE
12 CYCL CALL POS X+30 Y+20 Z+0 FMAX M3
13 CYCL CALL POS X+80 Y+50 Z+0 FMAX
```

3.3 DRILLING (Cycle 200)

Cycle run

1. The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the set-up clearance above the workpiece surface.
2. The tool drills to the first plunging depth at the programmed feed rate **F**.
3. The TNC returns the tool at **FMAX** to the set-up clearance, dwells there (if a dwell time was entered), and then moves at **FMAX** to the set-up clearance above the first plunging depth.
4. The tool then advances with another infeed at the programmed feed rate F
5. The TNC repeats this process (2 to 4) until the programmed depth is reached
6. The tool is retracted from the hole bottom to the set-up clearance or—if programmed—to the 2nd set-up clearance at **FMAX**.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

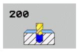
Danger of collision!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

Cycle parameters



▶ **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Enter a positive value. Input range 0 to 99999.9999; alternatively **PREDEF**

▶ **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole (tip of drill taper). Input range -99999.9999 to 99999.9999

▶ **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min during drilling. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**

▶ **Plunging depth** Q202 (incremental): Infeed per cut. Input range 0 to 99999.9999. The depth does not have to be a multiple of the plunging depth. The TNC will go to depth in one movement if:

- the plunging depth is equal to the depth
- the plunging depth is greater than the depth

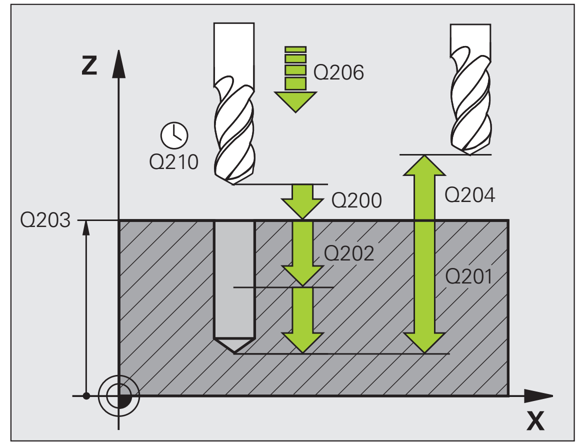
▶ **Dwell time at top** Q210: Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip removal. Input range 0 to 3600.0000; alternatively **PREDEF**

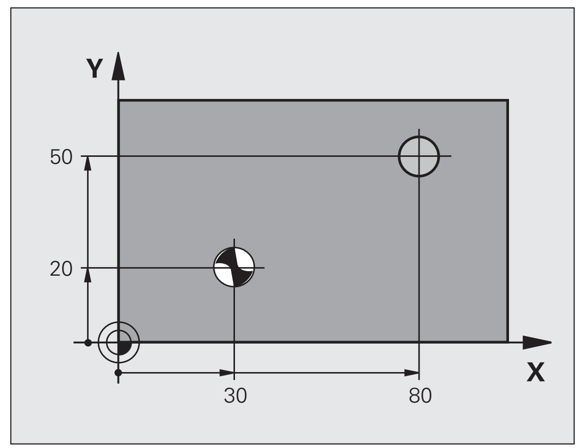
▶ **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

▶ **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**

▶ **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000; alternatively **PREDEF**

▶ **DEPTH REFERENCE** Q395: Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the TNC is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the T ANGLE column of the tool table TOOL.T.
0 = Depth referenced to tool tip
1 = Depth referenced to the cylindrical part of the tool





Example: NC blocks

```
11 CYCL DEF 200 DRILLING
    Q200=2     ;SET-UP CLEARANCE
    Q201=-15   ;DEPTH
    Q206=250   ;FEED RATE FOR PLNGNG
    Q202=5     ;PLUNGING DEPTH
    Q210=0     ;DWELL TIME AT TOP
    Q203=+20   ;SURFACE COORDINATE
    Q204=100   ;2ND SET-UP CLEARANCE
    Q211=0.1   ;DWELL TIME AT DEPTH
    Q395=0     ;DEPTH REFERENCE
12 L X+30 Y+20 FMAX M3 M99
14 L X+80 Y+50 FMAX M99
```

3.4 REAMING (Cycle 201, DIN/ISO: G201)

Cycle run

1. The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
2. The tool reams to the entered depth at the programmed feed rate **F**.
3. If programmed, the tool remains at the hole bottom for the entered dwell time.
4. The tool then retracts to the set-up clearance at the feed rate F, and from there—if programmed—to the 2nd set-up clearance at **FMAX**.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

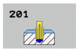
Danger of collision!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

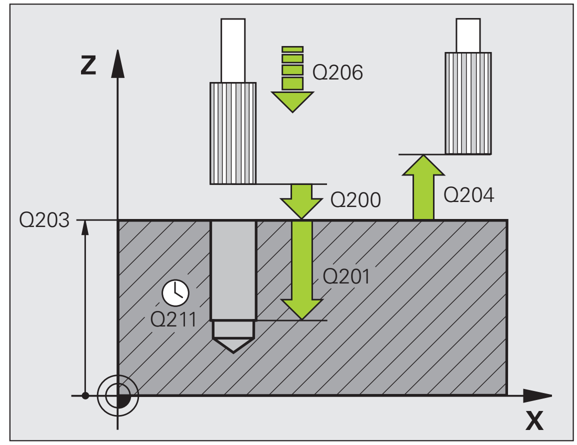
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

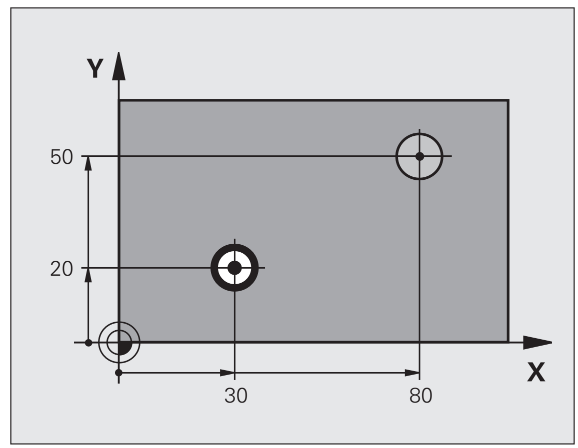
Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

Cycle parameters



- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min during reaming. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000; alternatively **PREDEF**
- **Feed rate for retraction** Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the feed rate for reaming applies. Input range 0 to 99999.999
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range 0 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**





Example: NC blocks

```
11 CYCL DEF 201 REAMING
    Q200=2     ;SET-UP CLEARANCE
    Q201=-15   ;DEPTH
    Q206=100   ;FEED RATE FOR PLNGNG
    Q211=0.5   ;DWELL TIME AT DEPTH
    Q208=250   ;RETRACTION FEED RATE
    Q203=+20   ;SURFACE COORDINATE
    Q204=100   ;2ND SET-UP CLEARANCE
12 L X+30 Y+20 FMAX M3
13 CYCL CALL
14 L X+80 Y+50 FMAX M9
15 L Z+100 FMAX M2
```

3.5 BORING (Cycle 202, DIN/ISO: G202)

Cycle run

1. The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the set-up clearance above the workpiece surface.
2. The tool bores to the programmed depth at the feed rate for plunging.
3. If programmed, the tool remains at the hole bottom for the entered dwell time with active spindle rotation for cutting free.
4. The TNC then orients the spindle to the position that is defined in parameter Q336.
5. If retraction is selected, the tool retracts in the programmed direction by 0.2 mm (fixed value)
6. The TNC moves the tool at the retraction feed rate to the set-up clearance and then, if entered, to the 2nd set-up clearance at **FMAX**. If Q214=0 the tool point remains on the wall of the hole.

Please note while programming:

Machine and TNC must be specially prepared by the machine tool builder for use of this cycle.

This cycle is effective only for machines with servo-controlled spindle.

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

After the cycle is completed, the TNC restores the coolant and spindle conditions that were active before the cycle call.

Danger of collision!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

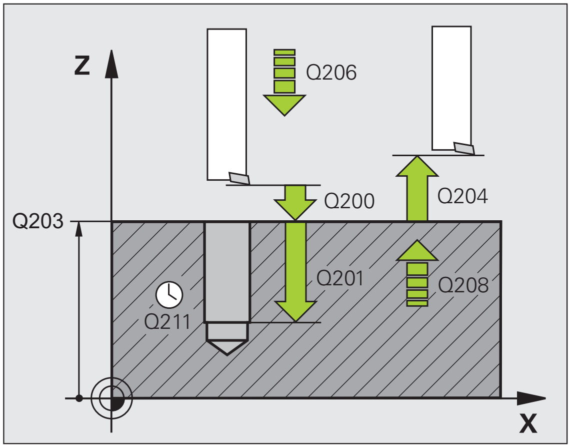
Select a disengaging direction in which the tool moves away from the edge of the hole.

Check the position of the tool tip when you program a spindle orientation to the angle that you enter in Q336 (for example, in the Positioning with Manual Data Input mode of operation). Set the angle so that the tool tip is parallel to a coordinate axis.

During retraction the TNC automatically takes an active rotation of the coordinate system into account.

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

Cycle parameters

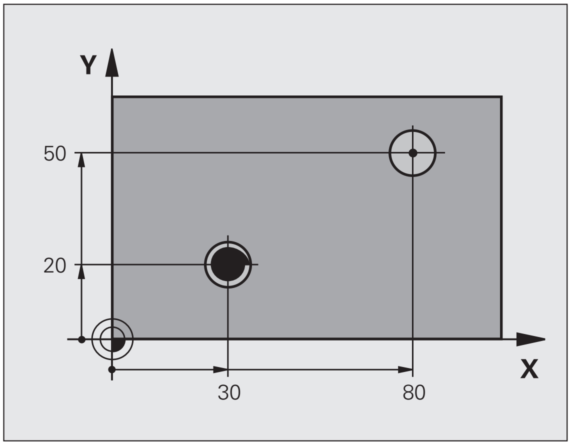


- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min during boring. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000; alternatively **PREDEF**
- **Feed rate for retraction** Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the feed rate for plunging applies. Input range 0 to 99999.999; alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.999; alternatively **PREDEF**

- **Disengaging direction (0/1/2/3/4)** Q214: Determine the direction in which the TNC retracts the tool at the hole bottom (after spindle orientation).

  0 Do not retract tool
  1 Retract tool in the negative ref. axis direction
  2 Retract tool in the negative minor axis direction
  3 Retract tool in the positive ref. axis direction
  4 Retract tool in the positive minor axis direction

- **Angle for spindle orientation** Q336 (absolute): Angle at which the TNC positions the tool before retracting it. Input range -360.000 to 360.000



Example:

```
10 L Z+100 R0 FMAX
11 CYCL DEF 202 BORING
    Q200=2     ;SET-UP CLEARANCE
    Q201=-15   ;DEPTH
    Q206=100   ;FEED RATE FOR PLNGNG
    Q211=0.5   ;DWELL TIME AT DEPTH
    Q208=250   ;RETRACTION FEED RATE
    Q203=+20   ;SURFACE COORDINATE
    Q204=100   ;2ND SET-UP CLEARANCE
    Q214=1     ;DISENGAGING DIRECTN
    Q336=0     ;ANGLE OF SPINDLE
12 L X+30 Y+20 FMAX M3
13 CYCL CALL
14 L X+80 Y+50 FMAX M99
```

3.6 UNIVERSAL DRILLING (Cycle 203, DIN/ISO: G203)

Cycle run

1 The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
2 The tool drills to the first plunging depth at the programmed feed rate **F**
3 If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool retracts at rapid traverse to the set-up clearance, remains there—if programmed—for the entered dwell time, and advances again at **FMAX** to the advanced stop distance Q256 above the currently drilled depth
4 The tool then advances with another infeed at the programmed feed rate. If programmed, the plunging depth is decreased after each infeed by the decrement.
5 The TNC repeats this process (2 to 4) until the programmed total hole depth is reached.
6 The tool remains at the hole bottom—if programmed—for the entered dwell time to cut free, and then retracts to the set-up clearance at the retraction feed rate. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Danger of collision!

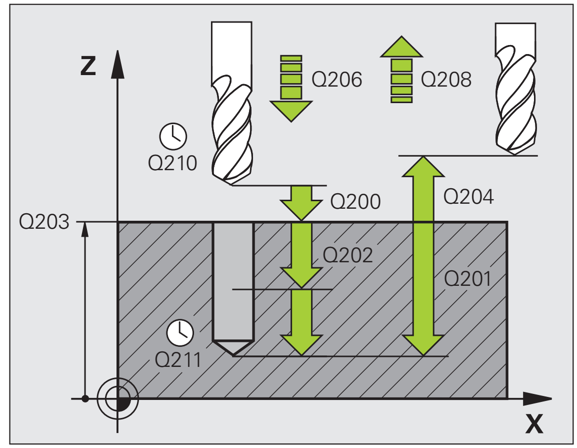
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

Cycle parameters

- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole (tip of drill taper). Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min during drilling. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Plunging depth** Q202 (incremental value): Infeed per cut. Input range 0 to 99999.9999. The depth does not have to be a multiple of the plunging depth. The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth and no chip breaking is defined
- **Dwell time at top** Q210: Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip removal. Input range 0 to 3600.0000; alternatively **PREDEF**
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Decrement** Q212 (incremental): Value by which the TNC decreases the plunging depth Q202 after each infeed. Input range 0 to 99999.9999



- **No. of breaks before retracting** Q213: Number of chip breaks after which the TNC is to withdraw the tool from the hole for chip removal. For chip breaking, the TNC retracts the tool each time by the value in Q256. Input range 0 to 99999
- **Minimum plunging depth** Q205 (incremental): If you have entered a decrement, the TNC limits the plunging depth to the value entered with Q205. Input range 0 to 99999.9999
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000; alternatively **PREDEF**
- **Feed rate for retraction** Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the TNC retracts the tool at the feed rate Q206. Input range 0 to 99999.999; alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Retraction rate for chip breaking** Q256 (incremental): Value by which the TNC retracts the tool during chip breaking. Input range 0.1000 to 99999.9999; alternatively **PREDEF**
- **DEPTH REFERENCE** Q395: Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the TNC is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the T ANGLE column of the tool table TOOL.T.
  0 = Depth referenced to tool tip
  1 = Depth referenced to the cylindrical part of the tool

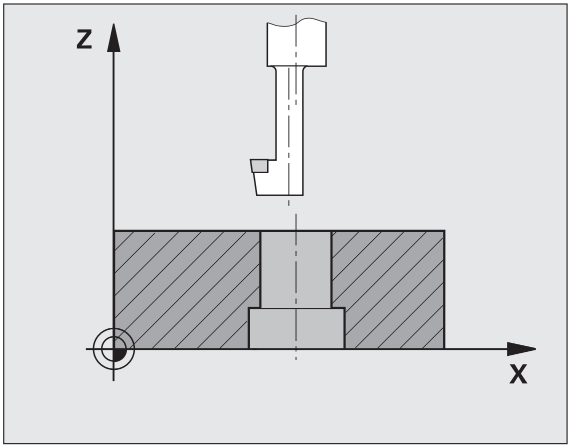
Example: NC blocks

```
11 CYCL DEF 203 UNIVERSAL DRILLING
    Q200=2     ;SET-UP CLEARANCE
    Q201=-20   ;DEPTH
    Q206=150   ;FEED RATE FOR PLNGNG
    Q202=5     ;PLUNGING DEPTH
    Q210=0     ;DWELL TIME AT TOP
    Q203=+20   ;SURFACE COORDINATE
    Q204=50    ;2ND SET-UP CLEARANCE
    Q212=0.2   ;DECREMENT
    Q213=3     ;NR OF BREAKS
    Q205=3     ;MIN. PLUNGING DEPTH
    Q211=0.25 ;DWELL TIME AT DEPTH
    Q208=500   ;RETRACTION FEED RATE
    Q256=0.2   ;DIST FOR CHIP BRKNG
    Q395=0     ;DEPTH REFERENCE
```

3.7 BACK BORING (Cycle 204, DIN/ISO: G204)

Cycle run

This cycle allows holes to be bored from the underside of the workpiece.



1 The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the set-up clearance above the workpiece surface.
2 The TNC then orients the spindle to the 0° position with an oriented spindle stop, and displaces the tool by the off-center distance.
3 The tool is then plunged into the already bored hole at the feed rate for pre-positioning until the tooth has reached the set-up clearance on the underside of the workpiece.
4 The TNC then centers the tool again over the bore hole, switches on the spindle and the coolant and moves at the feed rate for boring to the depth of bore.
5 If a dwell time is entered, the tool will pause at the top of the bore hole and will then be retracted from the hole again. Another oriented spindle stop is carried out and the tool is once again displaced by the off-center distance.
6 The TNC moves the tool at the pre-positioning feed rate to the set-up clearance and then, if entered, to the 2nd set-up clearance at **FMAX**.

Please note while programming:

Machine and TNC must be specially prepared by the machine tool builder for use of this cycle.

This cycle is effective only for machines with servo-controlled spindle.

Special boring bars for upward cutting are required for this cycle.

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter depth determines the working direction. Note: A positive sign bores in the direction of the positive spindle axis.

The entered tool length is the total length to the underside of the boring bar and not just to the tooth.

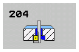
When calculating the starting point for boring, the TNC considers the tooth length of the boring bar and the thickness of the material.

You can also execute Cycle 204 with **M04** if you have programmed **M04** instead of **M03** prior to the cycle call.

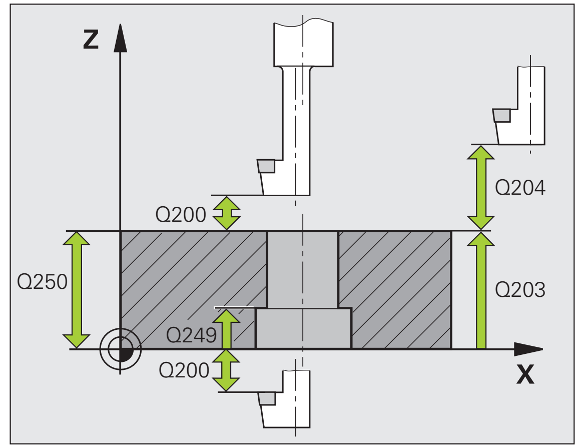
Danger of collision!

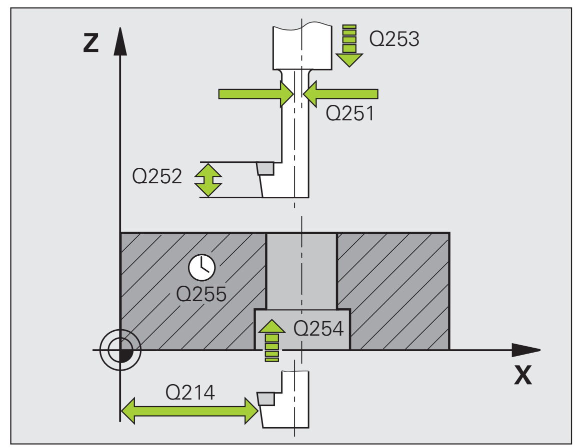
Check the position of the tool tip when you program a spindle orientation to the angle that you enter in **Q336** (for example, in the Positioning with Manual Data Input mode of operation). Set the angle so that the tool tip is parallel to a coordinate axis. Select a disengaging direction in which the tool moves away from the edge of the hole.

Cycle parameters



- **`Set-up clearance`** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **`PREDEF`**
- **`Depth of counterbore`** Q249 (incremental): Distance between underside of workpiece and the top of the hole. A positive sign means the hole will be bored in the positive spindle axis direction. Input range -99999.9999 to 99999.9999
- **`Material thickness`** Q250 (incremental): Thickness of the workpiece. Input range 0.0001 to 99999.9999
- **`Off-center distance`** Q251 (incremental): Off-center distance for the boring bar; value from tool data sheet. Input range 0.0001 to 99999.9999
- **`Tool edge height`** Q252 (incremental): Distance between the underside of the boring bar and the main cutting tooth; value from tool data sheet. Input range 0.0001 to 99999.9999
- **`Feed rate for pre-positioning`** Q253: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.999; alternatively **`FMAX`**, **`FAUTO`**, **`PREDEF`**
- **`Feed rate for countersinking`** Q254: Traversing speed of the tool in mm/min during countersinking. Input range 0 to 99999.999; alternatively **`FAUTO`**, **`FU`**
- **`Dwell time`** Q255: Dwell time in seconds at the top of the bore hole. Input range 0 to 3600.000





- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999; alternatively **PREDEF**
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Disengaging direction (0/1/2/3/4)** Q214: Determine the direction in which the TNC displaces the tool by the off-center distance (after spindle orientation). Input of 0 is not permitted

  1 Retract tool in the negative ref. axis direction
  2 Retract tool in the negative minor axis direction
  3 Retract tool in the positive ref. axis direction
  4 Retract tool in the positive minor axis direction
- **Angle for spindle orientation** Q336 (absolute): Angle at which the TNC positions the tool before it is plunged into or retracted from the bore hole. Input range -360.0000 to 360.0000

Example: NC blocks

```
11 CYCL DEF 204 BACK BORING
    Q200=2     ;SET-UP CLEARANCE
    Q249=+5    ;DEPTH OF COUNTERBORE
    Q250=20    ;MATERIAL THICKNESS
    Q251=3.5   ;OFF-CENTER DISTANCE
    Q252=15    ;TOOL EDGE HEIGHT
    Q253=750   ;F PRE-POSITIONING
    Q254=200   ;F COUNTERSINKING
    Q255=0     ;DWELL TIME
    Q203=+20   ;SURFACE COORDINATE
    Q204=50    ;2ND SET-UP CLEARANCE
    Q214=1     ;DISENGAGING DIRECTN
    Q336=0     ;ANGLE OF SPINDLE
```

3.8 UNIVERSAL PECKING (Cycle 205, DIN/ISO: G205)

Cycle run

1. The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
2. If you enter a deepened starting point, the TNC moves at the defined positioning feed rate to the set-up clearance above the deepened starting point.
3. The tool drills to the first plunging depth at the programmed feed rate **F**.
4. If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool retracts at rapid traverse to the set-up clearance, remains there—if programmed—for the entered dwell time, and advances again at **FMAX** to the advanced stop distance Q256 above the currently drilled depth
5. The tool then advances with another infeed at the programmed feed rate. If programmed, the plunging depth is decreased after each infeed by the decrement.
6. The TNC repeats this process (2 to 4) until the programmed total hole depth is reached.
7. The tool remains at the hole bottom—if programmed—for the entered dwell time to cut free, and then retracts to the set-up clearance at the retraction feed rate. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

If you enter different advanced stop distances for **Q258** and **Q259**, the TNC will change the advanced stop distances between the first and last plunging depths at the same rate.

If you use **Q379** to enter a deepened starting point, the TNC merely changes the starting point of the infeed movement. Retraction movements are not changed by the TNC, therefore they are calculated with respect to the coordinate of the workpiece surface.

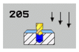
Danger of collision!

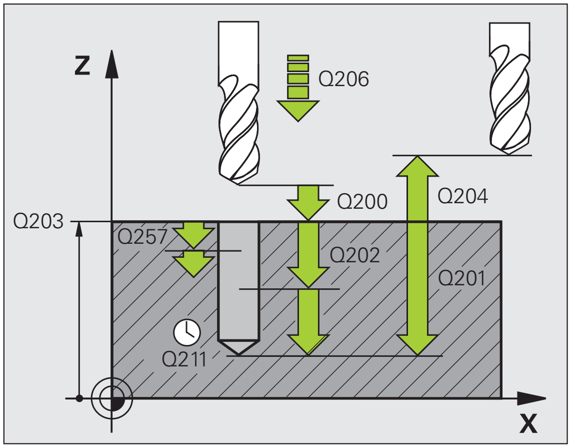
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

Cycle parameters





- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole (tip of drill taper). Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min during drilling. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Plunging depth** Q202 (incremental value): Infeed per cut. Input range 0 to 99999.9999. The depth does not have to be a multiple of the plunging depth. The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Decrement** Q212 (incremental): Value by which the TNC decreases the plunging depth Q202. Input range 0 to 99999.9999
- **Minimum plunging depth** Q205 (incremental): If you have entered a decrement, the TNC limits the plunging depth to the value entered with Q205. Input range 0 to 99999.9999
- **Upper advanced stop distance** Q258 (incremental): Set-up clearance for rapid traverse positioning when the TNC moves the tool again to the current plunging depth after retraction from the hole; value for the first plunging depth. Input range 0 to 99999.9999
- **Lower advanced stop distance** Q259 (incremental): Set-up clearance for rapid traverse positioning when the TNC moves the tool again to the current plunging depth after retraction from the hole; value for the last plunging depth. Input range 0 to 99999.9999

- **Infeed depth for chip breaking** Q257 (incremental): Depth at which the TNC carries out chip breaking. No chip breaking if 0 is entered. Input range 0 to 99999.9999
- **Retraction rate for chip breaking** Q256 (incremental): Value by which the TNC retracts the tool during chip breaking. The TNC retracts the tool at a feed rate of 3000 mm/min. Input range 0.1000 to 99999.9999; alternatively **PREDEF**
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000; alternatively **PREDEF**
- **Deepened starting point** Q379 (incremental with respect to the workpiece surface): Starting position of drilling if a shorter tool has already pilot drilled to a certain depth. The TNC moves at the **feed rate for pre-positioning** from the set-up clearance to the deepened starting point. Input range 0 to 99999.9999
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool in mm/min during positioning from set-up clearance to the deepened starting point. Effective only if the value entered for Q379 is not equal to 0. Input range 0 to 99999.999; alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Feed rate for retraction** Q208: Traversing speed of the tool in mm/min when retracting after the machining operation. If you enter Q208 = 0, the TNC retracts the tool at the feed rate Q206. Input range 0 to 99999.999; alternatively **FMAX**, **FAUTO**, **PREDEF**
- **DEPTH REFERENCE** Q395: Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the TNC is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the T ANGLE column of the tool table TOOL.T.
  0 = Depth referenced to tool tip
  1 = Depth referenced to the cylindrical part of the tool

Example: NC blocks

```
11 CYCL DEF 205 UNIVERSAL PECKING
    Q200=2     ;SET-UP CLEARANCE
    Q201=-80   ;DEPTH
    Q206=150   ;FEED RATE FOR PLNGNG
    Q202=15    ;PLUNGING DEPTH
    Q203=+100 ;SURFACE COORDINATE
    Q204=50    ;2ND SET-UP CLEARANCE
    Q212=0.5   ;DECREMENT
    Q205=3     ;MIN. PLUNGING DEPTH
    Q258=0.5   ;UPPER ADV. STOP DIST.
    Q259=1     ;LOWER ADV. STOP DIST.
    Q257=5     ;DEPTH FOR CHIP BRKNG
    Q256=0.2   ;DIST FOR CHIP BRKNG
    Q211=0.25 ;DWELL TIME AT DEPTH
    Q379=7.5   ;STARTING POINT
    Q253=750   ;F PRE-POSITIONING
    Q208=99999;RETRACTION FEED RATE
    Q395=0     ;DEPTH REFERENCE
```

3.9 BORE MILLING (Cycle 208)

Cycle run

1 The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the programmed set-up clearance above the workpiece surface and then moves the tool to the bore hole circumference on a rounded arc (if enough space is available).
2 The tool mills in a helix from the current position to the first plunging depth at the programmed feed rate **F**.
3 When the drilling depth is reached, the TNC once again traverses a full circle to remove the material remaining after the initial plunge.
4 The TNC then positions the tool at the center of the hole again.
5 Finally the TNC returns to the set-up clearance at **FMAX**. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

If you have entered the bore hole diameter to be the same as the tool diameter, the TNC will bore directly to the entered depth without any helical interpolation.

An active mirror function **does not** influence the type of milling defined in the cycle.

Note that if the infeed distance is too large, the tool or the workpiece may be damaged.

To prevent the infeeds from being too large, enter the maximum plunge angle of the tool in the **ANGLE** column of the tool table. The TNC then automatically calculates the max. infeed permitted and changes your entered value accordingly.

Danger of collision!

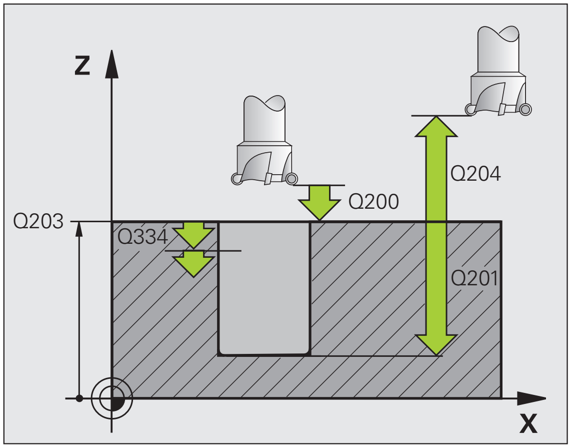
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

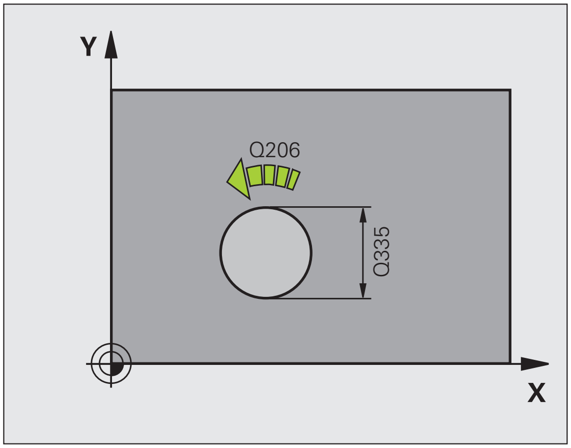
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

Cycle parameters

- **`Set-up clearance`** Q200 (incremental): Distance between tool lower edge and workpiece surface. Input range 0 to 99999.9999; alternatively **`PREDEF`**
- **`Depth`** Q201 (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **`Feed rate for plunging`** Q206: Traversing speed of the tool in mm/min during helical drilling. Input range 0 to 99999.999; alternatively **`FAUTO`**, **`FU`**, **`FZ`**
- **`Infeed per helix`** Q334 (incremental): Depth of the tool plunge with each helix (=360°). Input range 0 to 99999.9999
- **`Coordinate of workpiece surface`** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **`2nd set-up clearance`** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **`PREDEF`**
- **`Nominal diameter`** Q335 (absolute value): Bore-hole diameter. If you have entered the nominal diameter to be the same as the tool diameter, the TNC will bore directly to the entered depth without any helical interpolation. Input range 0 to 99999.9999
- **`Roughing diameter`** Q342 (absolute): As soon as you enter a value greater than 0 in Q342, the TNC no longer checks the ratio between the nominal diameter and the tool diameter. This allows you to rough-mill holes whose diameter is more than twice as large as the tool diameter. Input range 0 to 99999.9999
- **`Climb or up-cut`** Q351: Type of milling operation with M3
  +1 = climb milling
  –1 = up-cut milling
  `PREDEF` = use the default value from **`GLOBAL DEF`**





Example: NC blocks

```
12 CYCL DEF 208 BORE MILLING
    Q200=2     ;SET-UP CLEARANCE
    Q201=-80   ;DEPTH
    Q206=150   ;FEED RATE FOR PLNGNG
    Q334=1.5   ;PLUNGING DEPTH
    Q203=+100 ;SURFACE COORDINATE
    Q204=50    ;2ND SET-UP CLEARANCE
    Q335=25    ;NOMINAL DIAMETER
    Q342=0     ;ROUGHING DIAMETER
    Q351=+1    ;CLIMB OR UP-CUT
```

3.10 SINGLE-LIP DEEP-HOLE DRILLING (Cycle 241, DIN/ISO: G241)

Cycle run

1. The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
2. Then the TNC moves the tool at the defined positioning feed rate to the set-up clearance above the deepened starting point and switches on the drilling speed (**M3**) and the coolant. The approach motion is executed at the direction of rotation defined in the cycle, with clockwise, counterclockwise or stationary spindle.
3. The tool drills to the entered drilling depth or, if defined so, to the entered dwell depth, at the programmed feed rate **F**.
4. If programmed, the tool remains at the hole bottom for chip breaking. Then the TNC switches off the coolant and resets the drilling speed to the value defined for retraction.
5. After the dwell time at the hole bottom, the tool is retracted to the set-up clearance at the retraction feed rate. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

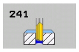
The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

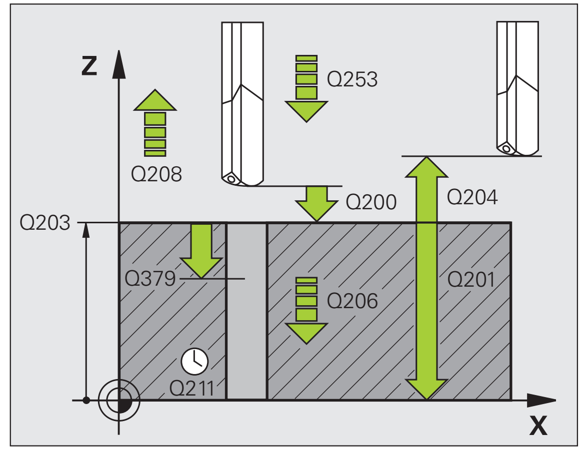
Danger of collision!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

Cycle parameters





- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min during drilling. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000; alternatively **PREDEF**
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Deepened starting point** Q379 (incremental with respect to the workpiece surface): Starting position for actual drilling operation. The TNC moves at the **feed rate for pre-positioning** from the set-up clearance to the deepened starting point. Input range 0 to 99999.9999
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool in mm/min during positioning from set-up clearance to the deepened starting point. Effective only if the value entered for Q379 is not equal to 0. Input range 0 to 99999.999; alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Feed rate for retraction** Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the TNC retracts the tool at the drilling feed rate Q206. Input range 0 to 99999.999; alternatively **FMAX**, **FAUTO**, **PREDEF**

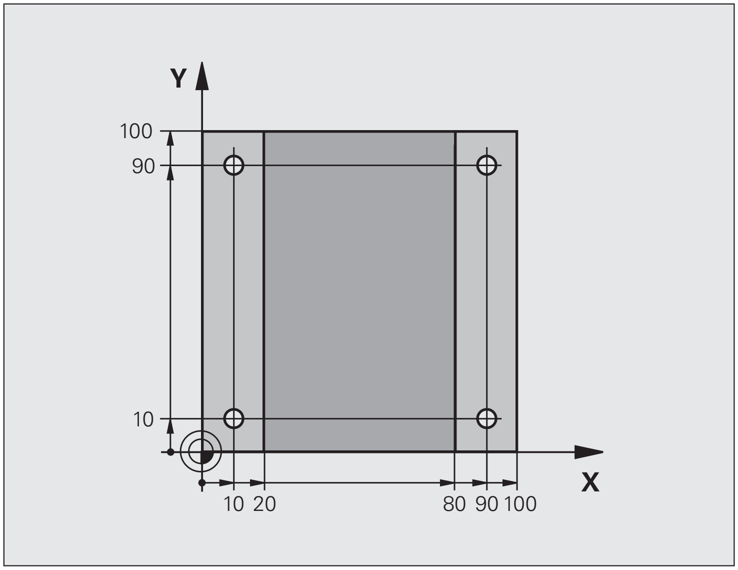
- **Rotat. dir. of entry/exit (3/4/5)** Q426: Desired direction of spindle rotation when tool moves into and retracts from the hole. Input range:
  3: Spindle rotation with M3
  4: Spindle rotation with M4
  5: Movement with stationary spindle
- **Spindle speed of entry/exit** Q427: Desired spindle speed when tool moves into and retracts from the hole. Input range 0 to 99999
- **Drilling speed** Q428: Desired speed for drilling. Input range 0 to 99999
- **M-codes: Coolant ON** Q429: M function for switching on the coolant. The TNC switches the coolant on if the tool is in the hole at the deepened starting point. Input range 0 to 999
- **M-codes: Coolant OFF** Q430: M function for switching off the coolant. The TNC switches the coolant off if the tool is at the hole depth. Input range 0 to 999
- **Dwell depth** Q435 (incremental): Coordinate in the spindle axis at which the tool is to dwell. If 0 is entered, the function is not active (standard setting) Application: During machining of through-holes some tools require a short dwell time before exiting the bottom of the hole in order to transport the chips to the top. Define a value smaller than the hole depth Q201; input range 0 to 99999.9999.

Example: NC blocks

```
11 CYCL DEF 241 SINGLE-LIP D.H.DRLNG
    Q200=2     ;SET-UP CLEARANCE
    Q201=-80   ;DEPTH
    Q206=150   ;FEED RATE FOR PLNGNG
    Q211=0.25  ;DWELL TIME AT DEPTH
    Q203=+100  ;SURFACE COORDINATE
    Q204=50    ;2ND SET-UP CLEARANCE
    Q379=7.5   ;STARTING POINT
    Q253=750   ;F PRE-POSITIONING
    Q208=1000  ;RETRACTION FEED RATE
    Q426=3     ;DIR. OF SPINDLE ROT.
    Q427=25    ;ROT. SPEED INFEED/OUT
    Q428=500   ;DRILLING SPEED
    Q429=8     ;COOLANT ON
    Q430=9     ;COOLANT OFF
    Q435=0     ;DWELL DEPTH
```

3.11 Programming examples

Example: Drilling cycles



0 BEGIN PGM C200 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S4500	Tool call (tool radius 3)
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 200 DRILLING	Cycle definition
Q200=2 ;SET-UP CLEARANCE	
Q201=-15 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=-10 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q211=0.2 ;DWELL TIME AT DEPTH	
Q395=0 ;DEPTH REFERENCE	

`6 L X+10 Y+10 R0 FMAX M3`	Approach hole 1, spindle ON
`7 CYCL CALL`	Cycle call
`8 L Y+90 R0 FMAX M99`	Approach hole 2, call cycle
`9 L X+90 R0 FMAX M99`	Approach hole 3, call cycle
`10 L Y+10 R0 FMAX M99`	Approach hole 4, call cycle
`11 L Z+250 R0 FMAX M2`	Retract the tool, end program
`12 END PGM C200 MM`	

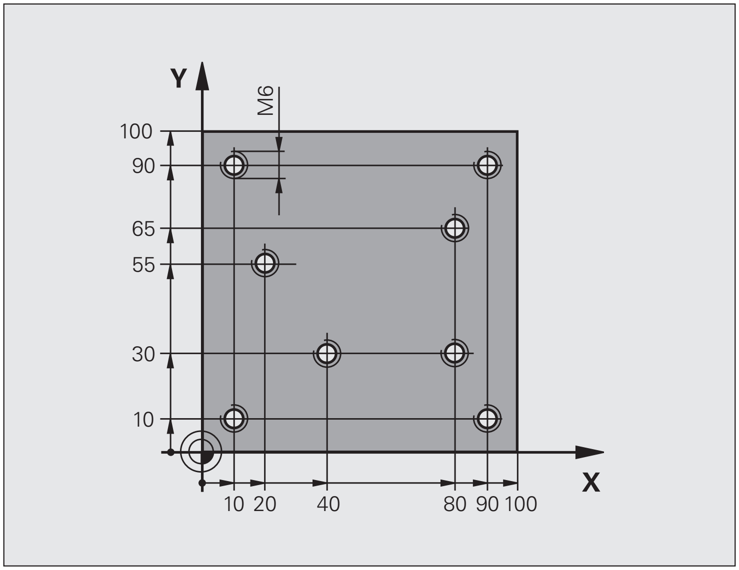
Example: Using drilling cycles in connection with PATTERN DEF

The drill hole coordinates are stored in the pattern definition **PATTERN DEF POS** and are called by the TNC with **CYCL CALL PAT**:

The tool radii are selected so that all work steps can be seen in the test graphics.

Program sequence

- Centering (tool radius 4)
- Drilling (tool radius 2.4)
- Tapping (tool radius 3)



0 BEGIN PGM 1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Y+0	
3 TOOL CALL 1 Z S5000	Call the centering tool (tool radius 4)
4 L Z+10 R0 F5000	Move tool to clearance height (enter a value for F): the TNC positions to the clearance height after every cycle
5 PATTERN DEF	Define all drilling positions in the point pattern
POS1(X+10 Y+10 Z+0)	
POS2(X+40 Y+30 Z+0)	
POS3(X+20 Y+55 Z+0)	
POS4(X+10 Y+90 Z+0)	
POS5(X+90 Y+90 Z+0)	
POS6(X+80 Y+65 Z+0)	
POS7(X+80 Y+30 Z+0)	
POS8(X+90 Y+10 Z+0)	

6 CYCL DEF 240 CENTERING	Cycle definition: CENTERING
Q200=2 ;SET-UP CLEARANCE	
Q343=0 ;SELECT DEPTH/DIA.	
Q201=-2 ;DEPTH	
Q344=-10 ;DIAMETER	
Q206=150 ;FEED RATE FOR PLNGNG	
Q211=0 ;DWELL TIME AT DEPTH	
Q203=+0 ;SURFACE COORDINATE	
Q204=50 ;2ND SET-UP CLEARANCE	
7 CYCL CALL PAT F5000 M13	Call the cycle in connection with the point pattern
8 L Z+100 R0 FMAX	Retract the tool, change the tool
9 TOOL CALL 2 Z S5000	Call the drilling tool (radius 2.4)
10 L Z+10 R0 F5000	Move tool to clearance height (enter a value for F)
11 CYCL DEF 200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-25 ;DEPTH	
Q206=150 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=50 ;2ND SET-UP CLEARANCE	
Q211=0.2 ;DWELL TIME AT DEPTH	
Q395=0 ;DEPTH REFERENCE	
12 CYCL CALL PAT F5000 M13	Call the cycle in connection with the point pattern
13 L Z+100 R0 FMAX	Retract the tool
14 TOOL CALL 3 Z S200	Call the tapping tool (radius 3)
15 L Z+50 R0 FMAX	Move tool to clearance height
16 CYCL DEF 206 TAPPING NEW	Cycle definition for tapping
Q200=2 ;SET-UP CLEARANCE	
Q201=-25 ;DEPTH OF THREAD	
Q206=150 ;FEED RATE FOR PLNGNG	
Q211=0 ;DWELL TIME AT DEPTH	
Q203=+0 ;SURFACE COORDINATE	
Q204=50 ;2ND SET-UP CLEARANCE	
17 CYCL CALL PAT F5000 M13	Call the cycle in connection with the hole pattern
18 L Z+100 R0 FMAX M2	Retract the tool, end program
19 END PGM 1 MM	

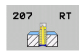
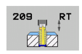
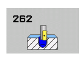
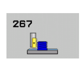
4

Fixed Cycles: Tapping / Thread Milling

4.1 Fundamentals

Overview

The TNC offers 8 cycles for all types of threading operations:

Cycle	Soft key	Page
206 TAPPING NEW With a floating tap holder, with automatic pre-positioning, 2nd set-up clearance		Page 107
207 RIGID TAPPING NEW Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance		Page 109
209 TAPPING W/ CHIP BREAKING Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance, chip breaking		Page 112
262 THREAD MILLING Cycle for milling a thread in pre-drilled material		Page 117
263 THREAD MILLING/CNTSNKG Cycle for milling a thread in pre-drilled material and machining a countersunk chamfer		Page 120
264 THREAD DRILLING/MILLING Cycle for drilling into solid material with subsequent milling of the thread with a tool		Page 124
265 HEL.THREAD DRILLING/MILLING Cycle for milling the thread into solid material		Page 128
267 OUTSIDE THREAD MILLING Cycle for milling an external thread and machining a countersunk chamfer		Page 128

4.2 TAPPING NEW with a Floating Tap Holder (Cycle 206, DIN/ISO: G206)

Cycle run

1. The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
2. The tool taps to the total hole depth in one movement.
3. Once the tool has reached the total hole depth, the direction of spindle rotation is reversed and the tool is retracted to the set-up clearance at the end of the dwell time. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.
4. At the set-up clearance, the direction of spindle rotation reverses once again.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

A floating tap holder is required for tapping. It must compensate the tolerances between feed rate and spindle speed during the tapping process.

When a cycle is being run, the spindle speed override knob is disabled. The feed-rate override knob is active only within a limited range, which is defined by the machine tool builder (refer to your machine manual).

For tapping right-hand threads activate the spindle with **M3**, for left-hand threads use **M4**.

If you enter the thread pitch of the tap in the **PITCH** column of the tool table, the TNC compares the thread pitch from the tool table with the thread pitch defined in the cycle. The TNC also displays an error message if the values do not match. In Cycle 206 the TNC uses the programmed rotational speed and the feed rate defined in the cycle to calculate the thread pitch.

Danger of collision!

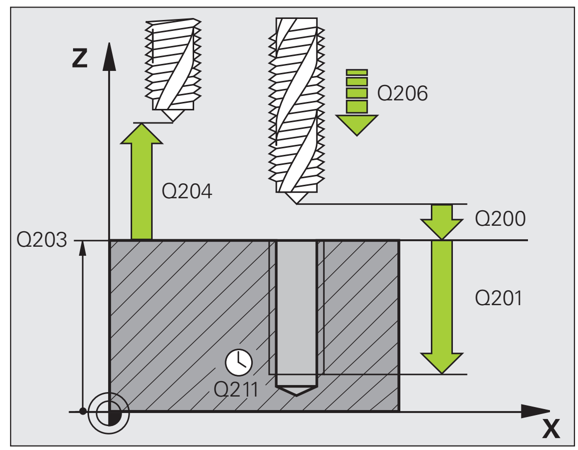
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

Cycle parameters

- **`Set-up clearance`** Q200 (incremental): Distance between tool tip (at starting position) and workpiece surface. Standard value: approx. 4 times the thread pitch. Input range 0 to 99999.9999; alternatively **`PREDEF`**
- **`Total hole depth`** Q201 (thread length, incremental): Distance between workpiece surface and end of thread. Input range -99999.9999 to 99999.9999
- **`Feed rate F`** Q206: Traversing speed of the tool during tapping. Input range 0 to 99999.999; alternatively **`FAUTO`**
- **`Dwell time at bottom`** Q211: Enter a value between 0 and 0.5 seconds to avoid wedging of the tool during retraction. Input range 0 to 3600.0000; alternatively **`PREDEF`**
- **`Coordinate of workpiece surface`** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **`2nd set-up clearance`** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **`PREDEF`**



The feed rate is calculated as follows: F = S x p

F: Feed rate (mm/min)
S: Spindle speed (rpm)
p: Thread pitch (mm)

Example: NC blocks

```
25 CYCL DEF 206 TAPPING NEW
     Q200=2    ;SET-UP CLEARANCE
     Q201=-20  ;DEPTH
     Q206=150  ;FEED RATE FOR PLNGNG
     Q211=0.25 ;DWELL TIME AT DEPTH
     Q203=+25  ;SURFACE COORDINATE
     Q204=50   ;2ND SET-UP CLEARANCE
```

Retracting after a program interruption

If you interrupt program run during tapping with the machine stop button, the TNC will display a soft key with which you can retract the tool.

4.3 RIGID TAPPING without a Floating Tap Holder NEW (Cycle 207, DIN/ISO: G207)

Cycle run

The TNC cuts the thread without a floating tap holder in one or more passes.

1. The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
2. The tool taps to the total hole depth in one movement.
3. Once the tool has reached the total hole depth, the direction of spindle rotation is reversed and the tool is retracted to the set-up clearance at the end of the dwell time. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.
4. The TNC brings the spindle to a stop at the set-up clearance.

Please note while programming:

Machine and TNC must be specially prepared by the machine tool builder for use of this cycle.

This cycle is effective only for machines with servo-controlled spindle.

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the total hole depth parameter determines the working direction.

The TNC calculates the feed rate from the spindle speed. If the spindle speed override is used during tapping, the feed rate is automatically adjusted.

The feed-rate override knob is disabled.

At the end of the cycle the spindle comes to a stop. Before the next operation, restart the spindle with **M3** (or **M4**).

If you enter the thread pitch of the tap in the **PITCH** column of the tool table, the TNC compares the thread pitch from the tool table with the thread pitch defined in the cycle. The TNC also displays an error message if the values do not match.

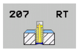
Danger of collision!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

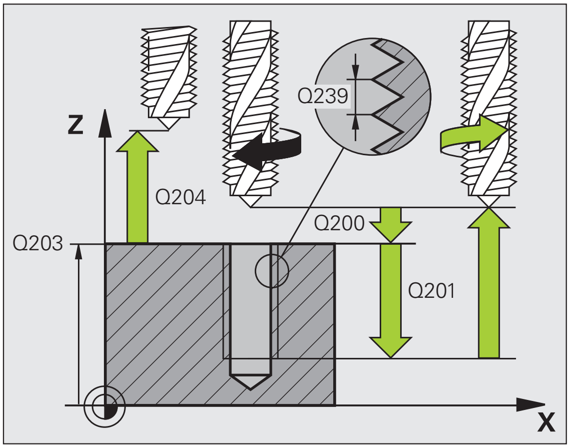
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

Cycle parameters



- **Set-up clearance** Q200 (incremental): Distance between tool tip (at starting position) and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Total hole depth** Q201 (incremental): Distance between workpiece surface and end of thread. Input range -99999.9999 to 99999.9999
- **Pitch** Q239
  Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  += right-hand thread
  –= left-hand thread
  Input range -99.9999 to 99.9999
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**



Retracting after a program interruption

If you interrupt program run during thread cutting with the machine stop button, the TNC will display the MANUAL OPERATION soft key. If you press the MANUAL OPERATION soft key, you can retract the tool under program control. Simply press the positive axis direction button of the active spindle axis.

Example: NC blocks

```
26 CYCL DEF 207 RIGID TAPPING NEW
     Q200=2     ;SET-UP CLEARANCE
     Q201=-20   ;DEPTH
     Q239=+1    ;PITCH
     Q203=+25   ;SURFACE COORDINATE
     Q204=50    ;2ND SET-UP CLEARANCE
```

4.4 TAPPING WITH CHIP BREAKING (Cycle 209, DIN/ISO: G209)

Cycle run

The TNC machines the thread in several passes until it reaches the programmed depth. You can define in a parameter whether the tool is to be retracted completely from the hole for chip breaking.

1. The TNC positions the tool in the tool axis at rapid traverse **FMAX** to the programmed set-up clearance above the workpiece surface. There it carries out an oriented spindle stop.
2. The tool moves to the programmed infeed depth, reverses the direction of spindle rotation and retracts by a specific distance or completely for chip release, depending on the definition. If you have defined a factor for increasing the spindle speed, the TNC retracts from the hole at the corresponding speed.
3. It then reverses the direction of spindle rotation again and advances to the next infeed depth.
4. The TNC repeats this process (2 to 3) until the programmed thread depth is reached.
5. The tool is then retracted to the set-up clearance. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.
6. The TNC brings the spindle to a stop at the set-up clearance.

Please note while programming:

Machine and TNC must be specially prepared by the machine tool builder for use of this cycle.

This cycle is effective only for machines with servo-controlled spindle.

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the parameter thread depth determines the working direction.

The TNC calculates the feed rate from the spindle speed. If the spindle speed override is used during tapping, the feed rate is automatically adjusted.

The feed-rate override knob is disabled.

If you defined an rpm factor for fast retraction in cycle parameter **Q403**, the TNC limits the speed to the maximum speed of the active gear range.

At the end of the cycle the spindle comes to a stop. Before the next operation, restart the spindle with **M3** (or **M4**).

If you enter the thread pitch of the tap in the **PITCH** column of the tool table, the TNC compares the thread pitch from the tool table with the thread pitch defined in the cycle. The TNC also displays an error message if the values do not match.

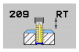
Danger of collision!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

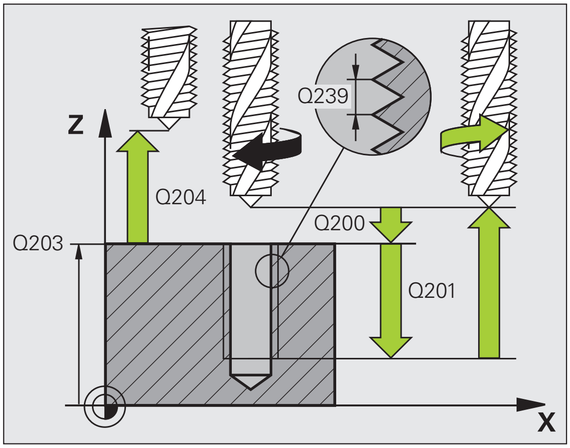
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

Cycle parameters



- **Set-up clearance** Q200 (incremental): Distance between tool tip (at starting position) and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Thread depth** Q201 (incremental): Distance between workpiece surface and end of thread. Input range -99999.9999 to 99999.9999
- **Pitch** Q239
  Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  += right-hand thread
  –= left-hand thread
  Input range -99.9999 to 99.9999
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Infeed depth for chip breaking** Q257 (incremental): Depth at which TNC carries out chip breaking. Input range 0 to 99999.9999
- **Retraction rate for chip breaking** Q256: The TNC multiplies the pitch Q239 by the programmed value and retracts the tool by the calculated value during chip breaking. If you enter Q256 = 0, the TNC retracts the tool completely from the hole (to the set-up clearance) for chip breaking. Input range 0 to 99999.9999
- **Angle for spindle orientation** Q336 (absolute): Angle at which the TNC positions the tool before machining the thread. This allows you to regroove the thread, if required. Input range -360.0000 to 360.0000
- **RPM factor for retraction** Q403: Factor by which the TNC increases the spindle speed—and therefore also the retraction feed rate—when retracting from the drill hole. Input range 0.0001 to 10; the speed is increased at most to the maximum speed of the active gear range.



Example: NC blocks

```
26 CYCL DEF 209 TAPPING W/ CHIP BRKG
    Q200=2     ;SET-UP CLEARANCE
    Q201=-20   ;DEPTH
    Q239=+1    ;PITCH
    Q203=+25   ;SURFACE COORDINATE
    Q204=50    ;2ND SET-UP CLEARANCE
    Q257=5     ;DEPTH FOR CHIP BRKNG
    Q256=+1    ;DIST FOR CHIP BRKNG
    Q336=50    ;ANGLE OF SPINDLE
    Q403=1.5   ;RPM FACTOR
```

Retracting after a program interruption

If you interrupt program run during thread cutting with the machine stop button, the TNC will display the MANUAL OPERATION soft key. If you press the MANUAL OPERATION soft key, you can retract the tool under program control. Simply press the positive axis direction button of the active spindle axis.

4.5 Fundamentals of thread milling

Requirements

- Your machine tool should feature internal spindle cooling (cooling lubricant at least 30 bars, compressed air supply at least 6 bars).
- Thread milling usually leads to distortions of the thread profile. To correct this effect, you need tool-specific compensation values which are given in the tool catalog or are available from the tool manufacturer. You program the compensation with the delta value for the tool radius **DR** in the **TOOL CALL**.
- The Cycles 262, 263, 264 and 267 can only be used with rightward rotating tools. For Cycle 265 you can use rightward and leftward rotating tools.
- The working direction is determined by the following input parameters: Algebraic sign Q239 (+ = right-hand thread / – = left-hand thread) and milling method Q351 (+1 = climb / –1 = up-cut). The table below illustrates the interrelation between the individual input parameters for rightward rotating tools.

Internal thread	Pitch	Climb/Up-cut	Work direction
Right-handed	+	+1(RL)	Z+
Left-handed	–	–1(RR)	Z+
Right-handed	+	–1(RR)	Z–
Left-handed	–	+1(RL)	Z–

External thread	Pitch	Climb/Up-cut	Work direction
Right-handed	+	+1(RL)	Z–
Left-handed	–	–1(RR)	Z–
Right-handed	+	–1(RR)	Z+
Left-handed	–	+1(RL)	Z+

The TNC references the programmed feed rate during thread milling to the tool cutting edge. Since the TNC, however, always displays the feed rate relative to the path of the tool tip, the displayed value does not match the programmed value.

The machining direction of the thread changes if you execute a thread milling cycle in connection with Cycle 8 MIRROR IMAGE in only one axis.

Danger of collision!

Always program the same algebraic sign for the infeeds: Cycles comprise several sequences of operation that are independent of each other. The order of precedence according to which the work direction is determined is described with the individual cycles. For example, if you only want to repeat the countersinking process of a cycle, enter 0 for the thread depth. The work direction will then be determined from the countersinking depth.

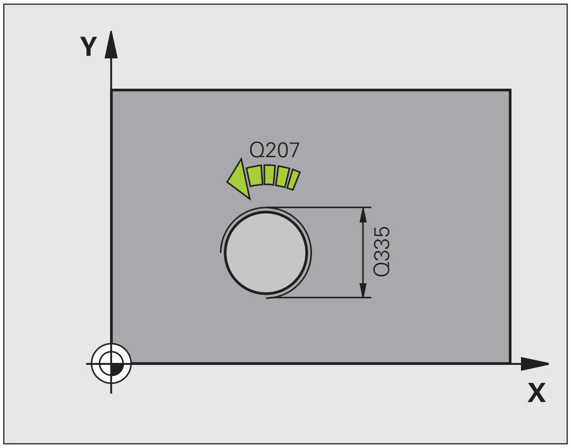
Procedure in case of a tool break

If a tool break occurs during thread cutting, stop program run, change to the Positioning with MDI operating mode and move the tool on a linear path to the hole center. You can then retract the tool in the infeed axis and replace it.

4.6 THREAD MILLING (Cycle 262, DIN/ISO: G262)

Cycle run

1. The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
2. The tool moves at the programmed feed rate for pre-positioning to the starting plane. The starting plane is derived from the algebraic sign of the thread pitch, the milling method (climb or up-cut milling) and the number of threads per step.
3. The tool then approaches the thread diameter tangentially in a helical movement. Before the helical approach, a compensating motion of the tool axis is carried out in order to begin at the programmed starting plane for the thread path.
4. Depending on the setting of the parameter for the number of threads, the tool mills the thread in one helical movement, in several offset helical movements or in one continuous helical movement.
5. After this, the tool departs the contour tangentially and returns to the starting point in the working plane.
6. At the end of the cycle, the TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance.



Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter "thread depth" determines the working direction. If you program the thread DEPTH = 0, the cycle will not be executed.

The nominal thread diameter is approached in a semi-circle from the center. A pre-positioning movement to the side is carried out if the tool diameter is smaller than the nominal thread diameter by four times the thread pitch.

Note that the TNC makes a compensation movement in the tool axis before the approach movement. The length of the compensation movement is at most half of the thread pitch. Ensure sufficient space in the hole!

If you change the thread depth, the TNC automatically changes the starting point for the helical movement.

Danger of collision!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

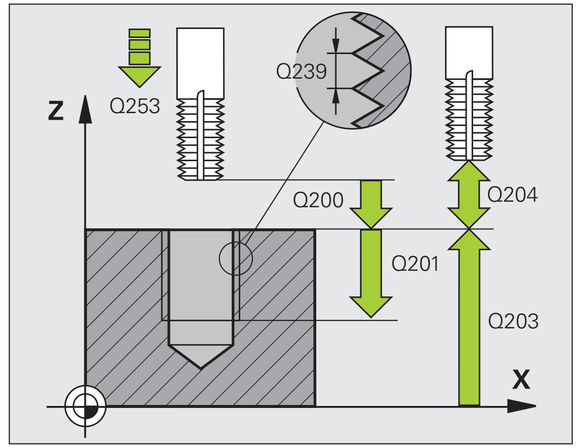
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

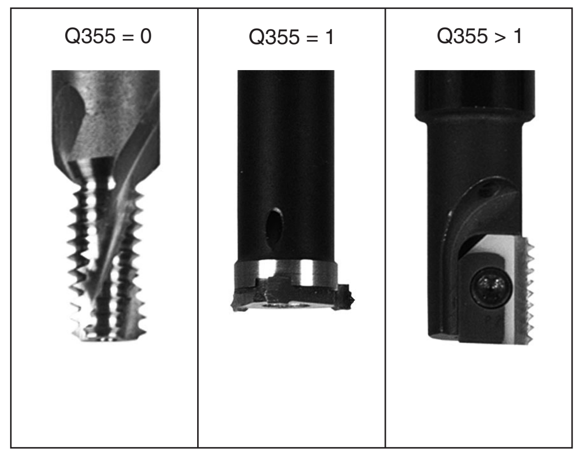
Keep in mind that if the depth is changed, the TNC adjust the starting angle so that the tool reaches the defined depth at the 0° position of the spindle. In such cases, recutting the thread may result in a second thread groove.

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

Cycle parameters

- **Nominal diameter** Q335: Nominal thread diameter. Input range 0 to 99999.9999
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  += right-hand thread
  – = left-hand thread
  Input range -99.9999 to 99.9999
- **Thread depth** Q201 (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- **Threads per step** Q355: Number of thread revolutions by which the tool is moved:
  0 = one 360° helical line to the thread depth
  1 = continuous helical path over the entire length of the thread
  >1 = several helical paths with approach and departure; between them, the TNC offsets the tool by Q355, multiplied by the pitch. Input range 0 to 99999
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.999; alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Climb or up-cut** Q351: Type of milling operation with M3
  +1 = climb milling
  –1 = up-cut milling
  Alternatively **PREDEF**
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999; alternatively **FAUTO**
- **Feed rate for approach** Q512: Traversing speed of the tool in mm/min during entry into the thread. Input range 0 to 99999.999; alternatively **FAUTO**





Example: NC blocks

```
25 CYCL DEF 262 THREAD MILLING
    Q335=10    ;NOMINAL DIAMETER
    Q239=+1.5  ;PITCH
    Q201=-20   ;DEPTH OF THREAD
    Q355=0     ;THREADS PER STEP
    Q253=750   ;F PRE-POSITIONING
    Q351=+1    ;CLIMB OR UP-CUT
    Q200=2     ;SET-UP CLEARANCE
    Q203=+30   ;SURFACE COORDINATE
    Q204=50    ;2ND SET-UP CLEARANCE
    Q207=500   ;FEED RATE FOR MILLING
    Q512=50    ;FEED RATE FOR APPROACH
```

4.7 THREAD MILLING/COUNTERSINKING (Cycle 263, DIN/ISO: G263)

Cycle run

1 The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.

Countersinking

2 The tool moves at the feed rate for pre-positioning to the countersinking depth minus the set-up clearance, and then at the feed rate for countersinking to the countersinking depth.

3 If a set-up clearance to the side has been entered, the TNC immediately positions the tool at the feed rate for pre-positioning to the countersinking depth.

4 Then, depending on the available space, the TNC makes a tangential approach to the core diameter, either tangentially from the center or with a pre-positioning move to the side, and follows a circular path.

Countersinking at front

5 The tool moves at the feed rate for pre-positioning to the countersinking depth at front.

6 The TNC positions the tool without compensation from the center on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking.

7 The tool then moves in a semicircle to the hole center.

Thread milling

8 The TNC moves the tool at the programmed feed rate for pre-positioning to the starting plane for the thread. The starting plane is determined from the algebraic sign of the thread pitch and the type of milling (climb or up-cut).

9 Then the tool moves tangentially on a helical path to the thread diameter and mills the thread with a 360° helical motion.

10 After this, the tool departs the contour tangentially and returns to the starting point in the working plane.

11 At the end of the cycle, the TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance.

Please note while programming:

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign of the cycle parameters thread depth, countersinking depth or depth at front determines the working direction. The working direction is defined in the following sequence:
1 Thread depth
2 Countersinking depth
3 Depth at front

If you program a depth parameter to be 0, the TNC does not execute that step.

If you want to countersink at front, define the countersinking depth as 0.

Program the thread depth as a value smaller than the countersinking depth by at least one-third the thread pitch.

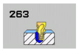
Danger of collision!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

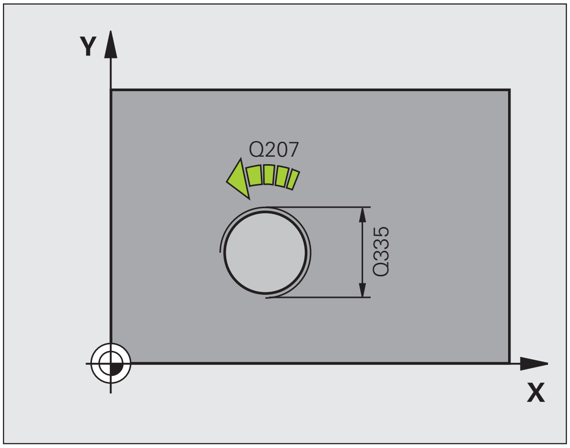
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

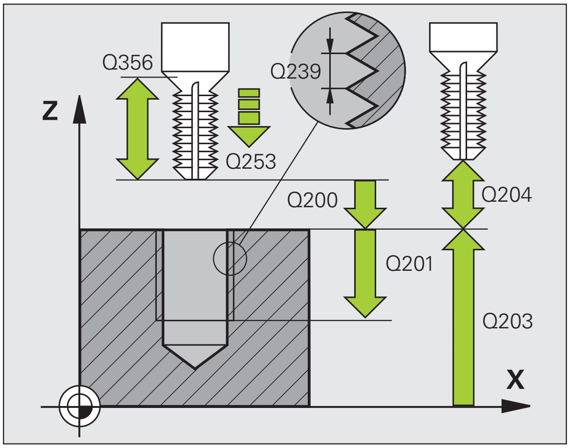
Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

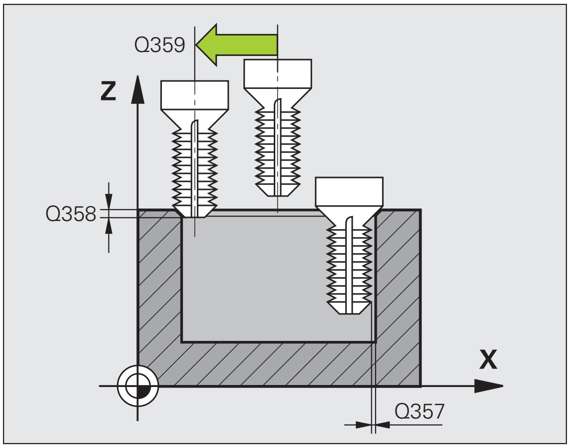
Cycle parameters



- **`Nominal diameter`** Q335: Nominal thread diameter. Input range 0 to 99999.9999
- **`Thread pitch`** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  += right-hand thread
  – = left-hand thread
  Input range -99.9999 to 99.9999
- **`Thread depth`** Q201 (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- **`Countersinking depth`** Q356 (incremental): Distance between tool tip and the top surface of the workpiece. Input range -99999.9999 to 99999.9999
- **`Feed rate for pre-positioning`** Q253: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.999; alternatively **`FMAX`**, **`FAUTO`**, **`PREDEF`**
- **`Climb or up-cut`** Q351: Type of milling operation with M3
  +1 = climb milling
  –1 = up-cut milling
  Alternatively **`PREDEF`**
- **`Set-up clearance`** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **`PREDEF`**
- **`Set-up clearance to the side`** Q357 (incremental): Distance between tool tooth and the wall of the hole. Input range 0 to 99999.9999
- **`Depth at front`** Q358 (incremental): Distance between tool tip and the top surface of the workpiece for countersinking at front. Input range -99999.9999 to 99999.9999
- **`Countersinking offset at front`** Q359 (incremental): Distance by which the TNC moves the tool center away from the hole center. Input range 0 to 99999.9999







- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Feed rate for countersinking** Q254: Traversing speed of the tool in mm/min during countersinking. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.9999; alternatively **FAUTO**
- **Feed rate for approach** Q512: Traversing speed of the tool in mm/min during entry into the thread. Input range 0 to 99999.999; alternatively **FAUTO**

Example: NC blocks

```
25 CYCL DEF 263 THREAD MLLNG/CNTSNKG
    Q335=10    ;NOMINAL DIAMETER
    Q239=+1.5  ;PITCH
    Q201=-16   ;DEPTH OF THREAD
    Q356=-20   ;COUNTERSINKING DEPTH
    Q253=750   ;F PRE-POSITIONING
    Q351=+1    ;CLIMB OR UP-CUT
    Q200=2     ;SET-UP CLEARANCE
    Q357=0.2   ;CLEARANCE TO SIDE
    Q358=+0    ;DEPTH AT FRONT
    Q359=+0    ;OFFSET AT FRONT
    Q203=+30   ;SURFACE COORDINATE
    Q204=50    ;2ND SET-UP CLEARANCE
    Q254=150   ;F COUNTERSINKING
    Q207=500   ;FEED RATE FOR MILLING
    Q512=50    ;FEED RATE FOR APPROACH
```

4.8 THREAD DRILLING/MILLING (Cycle 264, DIN/ISO: G264)

Cycle run

1 The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.

Drilling

2 The tool drills to the first plunging depth at the programmed feed rate for plunging.
3 If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool is moved at rapid traverse to the set-up clearance, and then at **FMAX** to the entered advanced stop distance above the first plunging depth.
4 The tool then advances with another infeed at the programmed feed rate.
5 The TNC repeats this process (2 to 4) until the programmed total hole depth is reached.

Countersinking at front

6 The tool moves at the feed rate for pre-positioning to the countersinking depth at front.
7 The TNC positions the tool without compensation from the center on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking.
8 The tool then moves in a semicircle to the hole center.

Thread milling

9 The TNC moves the tool at the programmed feed rate for pre-positioning to the starting plane for the thread. The starting plane is determined from the algebraic sign of the thread pitch and the type of milling (climb or up-cut).
10 Then the tool moves tangentially on a helical path to the thread diameter and mills the thread with a 360° helical motion.
11 After this, the tool departs the contour tangentially and returns to the starting point in the working plane.
12 At the end of the cycle, the TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign of the cycle parameters thread depth, countersinking depth or depth at front determines the working direction. The working direction is defined in the following sequence:
1 Thread depth
2 Total hole depth
3 Depth at front

If you program a depth parameter to be 0, the TNC does not execute that step.

Program the thread depth as a value smaller than the total hole depth by at least one-third the thread pitch.

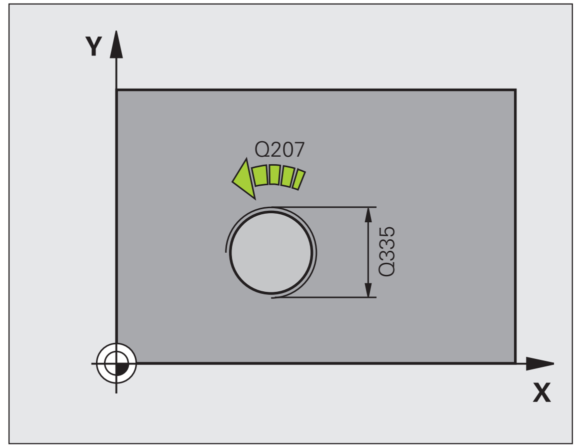
Danger of collision!

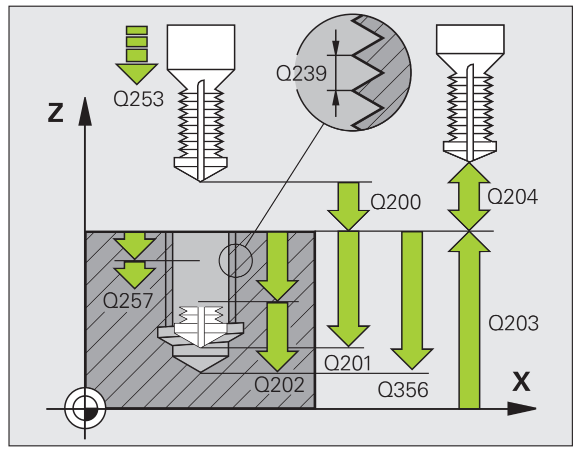
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

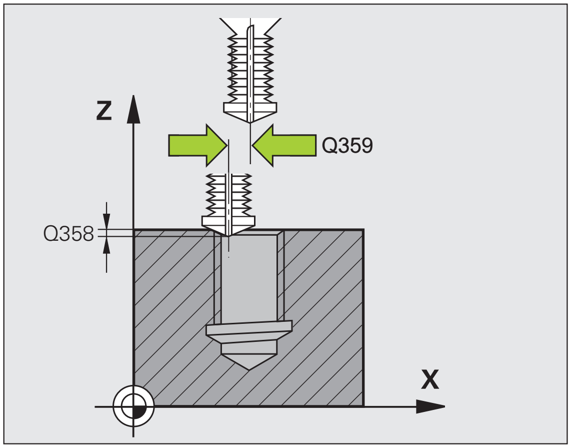
Cycle parameters





- **`Nominal diameter`** Q335: Nominal thread diameter. Input range 0 to 99999.9999
- **`Thread pitch`** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  += right-hand thread
  – = left-hand thread
  Input range -99.9999 to 99.9999
- **`Thread depth`** Q201 (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- **`Total hole depth`** Q356 (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **`Feed rate for pre-positioning`** Q253: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.999; alternatively **`FMAX`**, **`FAUTO`**, **`PREDEF`**
- **`Climb or up-cut`** Q351: Type of milling operation with M3
  +1 = climb milling
  –1 = up-cut milling
  Alternatively **`PREDEF`**
- **`Plunging depth`** Q202 (incremental value): Infeed per cut. The depth does not have to be a multiple of the plunging depth. Input range 0 to 99999.9999. The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **`Upper advanced stop distance`** Q258 (incremental): Set-up clearance for rapid traverse positioning when the TNC moves the tool again to the current plunging depth after retraction from the hole. Input range 0 to 99999.9999
- **`Infeed depth for chip breaking`** Q257 (incremental): Depth at which TNC carries out chip breaking. No chip breaking if 0 is entered. Input range 0 to 99999.9999; alternatively **`PREDEF`**
- **`Retraction rate for chip breaking`** Q256 (incremental): Value by which the TNC retracts the tool during chip breaking. Input range 0.1000 to 99999.9999

- **Depth at front** Q358 (incremental): Distance between tool tip and the top surface of the workpiece for countersinking at front. Input range -99999.9999 to 99999.9999
- **Countersinking offset at front** Q359 (incremental): Distance by which the TNC moves the tool center away from the hole center. Input range 0 to 99999.9999
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min during drilling. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.9999; alternatively **FAUTO**
- **Feed rate for approach** Q512: Traversing speed of the tool in mm/min during entry into the thread. Input range 0 to 99999.999; alternatively **FAUTO**



Example: NC blocks

```
25 CYCL DEF 264 THREAD DRILLNG/MLLNG
    Q335=10    ;NOMINAL DIAMETER
    Q239=+1.5  ;PITCH
    Q201=-16   ;DEPTH OF THREAD
    Q356=-20   ;TOTAL HOLE DEPTH
    Q253=750   ;F PRE-POSITIONING
    Q351=+1    ;CLIMB OR UP-CUT
    Q202=5     ;PLUNGING DEPTH
    Q258=0.2   ;ADVANCED STOP DISTANCE
    Q257=5     ;DEPTH FOR CHIP BRKNG
    Q256=0.2   ;DIST FOR CHIP BRKNG
    Q358=+0    ;DEPTH AT FRONT
    Q359=+0    ;OFFSET AT FRONT
    Q200=2     ;SET-UP CLEARANCE
    Q203=+30   ;SURFACE COORDINATE
    Q204=50    ;2ND SET-UP CLEARANCE
    Q206=150   ;FEED RATE FOR PLNGNG
    Q207=500   ;FEED RATE FOR MILLING
    Q512=50    ;FEED RATE FOR APPROACH
```

4.9 HELICAL THREAD DRILLING/MILLING (Cycle 265, DIN/ISO: G265)

Cycle run

1 The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.

Countersinking at front

2 If countersinking is before thread milling, the tool moves at the feed rate for countersinking to the sinking depth at front. If countersinking occurs after thread milling, the TNC moves the tool to the countersinking depth at the feed rate for pre-positioning.
3 The TNC positions the tool without compensation from the center on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking.
4 The tool then moves in a semicircle to the hole center.

Thread milling

5 The tool moves at the programmed feed rate for pre-positioning to the starting plane for the thread.
6 The tool then approaches the thread diameter tangentially in a helical movement.
7 The tool moves on a continuous helical downward path until it reaches the thread depth.
8 After this, the tool departs the contour tangentially and returns to the starting point in the working plane.
9 At the end of the cycle, the TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign of the cycle parameters thread depth or depth at front determines the working direction. The working direction is defined in the following sequence:
1 Thread depth
2 Depth at front

If you program a depth parameter to be 0, the TNC does not execute that step.

If you change the thread depth, the TNC automatically changes the starting point for the helical movement.

The type of milling (up-cut/climb) is determined by the thread (right-hand/left-hand) and the direction of tool rotation, since it is only possible to work in the direction of the tool.

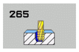
Danger of collision!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

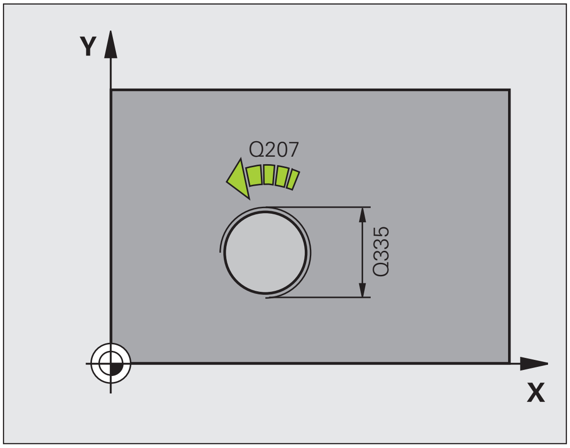
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

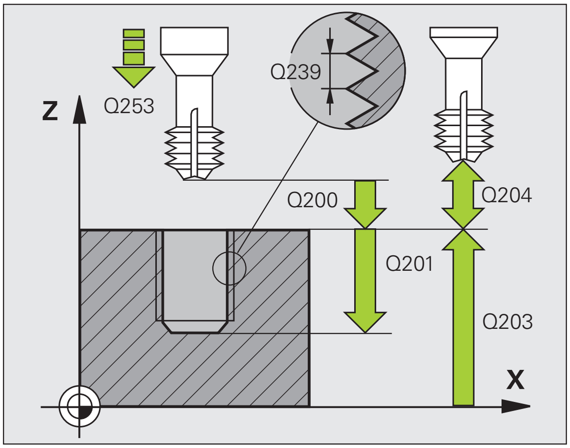
Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

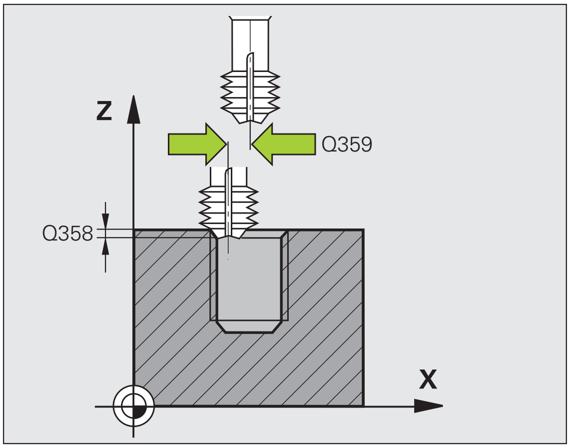
Cycle parameters



- **`Nominal diameter`** Q335: Nominal thread diameter. Input range 0 to 99999.9999
- **`Thread pitch`** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
  Input range -99.9999 to 99.9999
- **`Thread depth`** Q201 (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- **`Feed rate for pre-positioning`** Q253: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.999; alternatively **`FMAX`**, **`FAUTO`**, **`PREDEF`**
- **`Depth at front`** Q358 (incremental): Distance between tool tip and the top surface of the workpiece for countersinking at front. Input range -99999.9999 to 99999.9999
- **`Countersinking offset at front`** Q359 (incremental): Distance by which the TNC moves the tool center away from the hole center. Input range 0 to 99999.9999
- **`Countersink`** Q360: Execution of the chamfer
  0 = before thread machining
  1 = after thread machining
- **`Set-up clearance`** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **`PREDEF`**







- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Feed rate for countersinking** Q254: Traversing speed of the tool in mm/min during countersinking. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999; alternatively **FAUTO**

Example: NC blocks

```
25 CYCL DEF 265 HEL. THREAD DRLG/MLG
    Q335=10    ;NOMINAL DIAMETER
    Q239=+1.5  ;PITCH
    Q201=-16   ;DEPTH OF THREAD
    Q253=750   ;F PRE-POSITIONING
    Q358=+0    ;DEPTH AT FRONT
    Q359=+0    ;OFFSET AT FRONT
    Q360=0     ;COUNTERSINK
    Q200=2     ;SET-UP CLEARANCE
    Q203=+30   ;SURFACE COORDINATE
    Q204=50    ;2ND SET-UP CLEARANCE
    Q254=150   ;F COUNTERSINKING
    Q207=500   ;FEED RATE FOR MILLING
```

4.10 OUTSIDE THREAD MILLING (Cycle 267, DIN/ISO: G267)

Cycle run

1 The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.

Countersinking at front

2 The TNC moves on the reference axis of the working plane from the center of the stud to the starting point for countersinking at front. The position of the starting point is determined by the thread radius, tool radius and pitch.
3 The tool moves at the feed rate for pre-positioning to the countersinking depth at front.
4 The TNC positions the tool without compensation from the center on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking.
5 The tool then moves on a semicircle to the starting point.

Thread milling

6 The TNC positions the tool to the starting point if there has been no previous countersinking at front. Starting point for thread milling = starting point for countersinking at front.
7 The tool moves at the programmed feed rate for pre-positioning to the starting plane. The starting plane is derived from the algebraic sign of the thread pitch, the milling method (climb or up-cut milling) and the number of threads per step.
8 The tool then approaches the thread diameter tangentially in a helical movement.
9 Depending on the setting of the parameter for the number of threads, the tool mills the thread in one helical movement, in several offset helical movements or in one continuous helical movement.
10 After this, the tool departs the contour tangentially and returns to the starting point in the working plane.
11 At the end of the cycle, the TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance.

Please note while programming:

Program a positioning block for the starting point (stud center) in the working plane with radius compensation **R0**.

The offset required before countersinking at the front should be determined ahead of time. You must enter the value from the center of the stud to the center of the tool (uncorrected value).

The algebraic sign of the cycle parameters depth of thread or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
1 Thread depth
2nd Depth at front

If you program a depth parameter to be 0, the TNC does not execute that step.

The algebraic sign for the cycle parameter thread depth determines the working direction.

Danger of collision!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

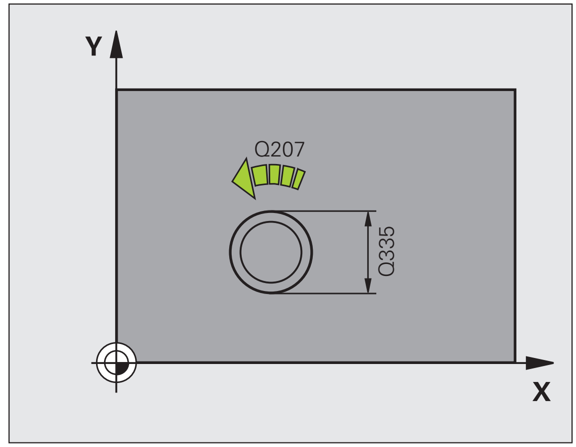
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

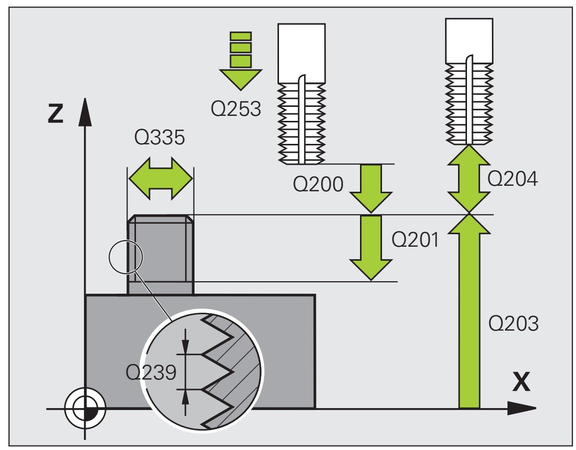
Keep in mind that if the depth is changed, the TNC adjust the starting angle so that the tool reaches the defined depth at the 0° position of the spindle. In such cases, recutting the thread may result in a second thread groove.

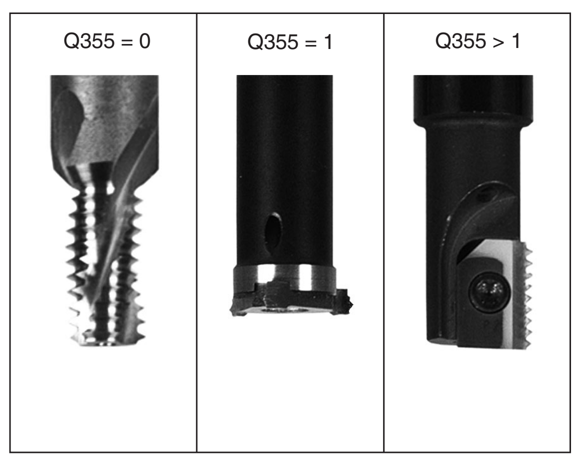
Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

Cycle parameters

- **`Nominal diameter`** Q335: Nominal thread diameter. Input range 0 to 99999.9999
- **`Thread pitch`** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  += right-hand thread
  – = left-hand thread
  Input range -99.9999 to 99.9999
- **`Thread depth`** Q201 (incremental): Distance between workpiece surface and root of thread
- **`Threads per step`** Q355: Number of thread revolutions by which the tool is moved:
  0 = one helical line to the thread depth
  1 = continuous helical path over the entire length of the thread
  >1 = several helical paths with approach and departure; between them, the TNC offsets the tool by Q355, multiplied by the pitch. Input range 0 to 99999
- **`Feed rate for pre-positioning`** Q253: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.999; alternatively **`FMAX`**, **`FAUTO`**, **`PREDEF`**
- **`Climb or up-cut`** Q351: Type of milling operation with M3
  +1 = climb milling
  –1 = up-cut milling
  Alternatively **`PREDEF`**







- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Depth at front** Q358 (incremental): Distance between tool tip and the top surface of the workpiece for countersinking at front. Input range -99999.9999 to 99999.9999
- **Countersinking offset at front** Q359 (incremental): Distance by which the TNC moves the tool center away from the stud center. Input range 0 to 99999.9999
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Feed rate for countersinking** Q254: Traversing speed of the tool in mm/min during countersinking. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999; alternatively **FAUTO**
- **Feed rate for approach** Q512: Traversing speed of the tool in mm/min during entry into the thread. Input range 0 to 99999.999; alternatively **FAUTO**

Example: NC blocks

```
25 CYCL DEF 267 OUTSIDE THREAD MLLNG
    Q335=10    ;NOMINAL DIAMETER
    Q239=+1.5  ;PITCH
    Q201=-20   ;DEPTH OF THREAD
    Q355=0     ;THREADS PER STEP
    Q253=750   ;F PRE-POSITIONING
    Q351=+1    ;CLIMB OR UP-CUT
    Q200=2     ;SET-UP CLEARANCE
    Q358=+0    ;DEPTH AT FRONT
    Q359=+0    ;OFFSET AT FRONT
    Q203=+30   ;SURFACE COORDINATE
    Q204=50    ;2ND SET-UP CLEARANCE
    Q254=150   ;F COUNTERSINKING
    Q207=500   ;FEED RATE FOR MILLING
    Q512=50    ;FEED RATE FOR APPROACH
```

4.11 Programming examples

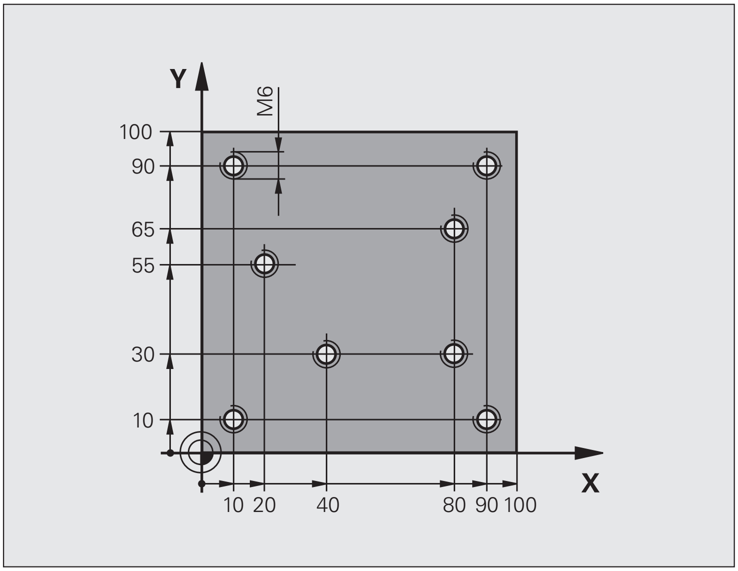
Example: Tapping

The drill hole coordinates are stored in the point table TAB1.PNT and are called by the TNC with **CYCL CALL PAT**.

The tool radii are selected so that all work steps can be seen in the test graphics.

Program sequence

- Centering
- Drilling
- Tapping

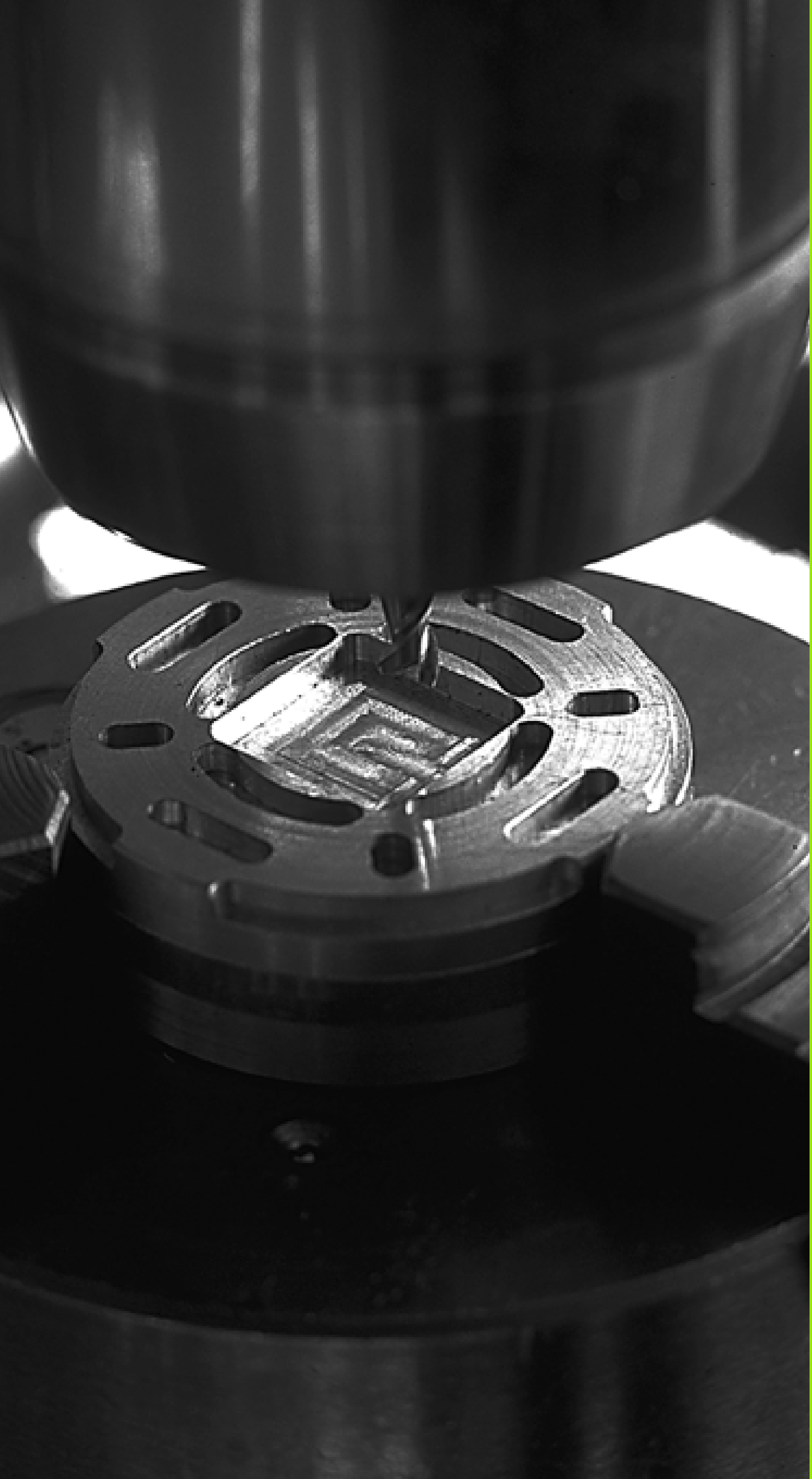


0 BEGIN PGM 1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Y+0	
3 TOOL DEF 1 L+0 R+4	Tool definition of centering tool
4 TOOL DEF 2 L+0 2.4	Tool definition of drill
5 TOOL DEF 3 L+0 R+3	Tool definition of tap
6 TOOL CALL 1 Z S5000	Call tool: centering drill
7 L Z+10 R0 F5000	Move tool to clearance height (enter a value for F): the TNC positions to the clearance height after every cycle
8 SEL PATTERN "TAB1"	Definition of point table
9 CYCL DEF 200 DRILLING	Cycle definition: CENTERING
Q200=2 ;SET-UP CLEARANCE	
Q201=-2 ;DEPTH	
Q206=150 ;FEED RATE FOR PLNGNG	
Q202=2 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	0 must be entered here, effective as defined in point table

Q204=0 ;2ND SET-UP CLEARANCE	0 must be entered here, effective as defined in point table
Q211=0.2 ;DWELL TIME AT DEPTH	
Q395=0 ;DEPTH REFERENCE	
10 CYCL CALL PAT F5000 M3	Cycle call in connection with point table TAB1.PNT, feed rate between the points: 5000 mm/min
11 L Z+100 R0 FMAX M6	Retract the tool, change the tool
12 TOOL CALL 2 Z S5000	Call tool: drill
13 L Z+10 R0 F5000	Move tool to clearance height (enter a value for F)
14 CYCL DEF 200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-25 ;DEPTH	
Q206=150 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	0 must be entered here, effective as defined in point table
Q204=0 ;2ND SET-UP CLEARANCE	0 must be entered here, effective as defined in point table
Q211=0.2 ;DWELL TIME AT DEPTH	
Q395=0 ;DEPTH REFERENCE	
15 CYCL CALL PAT F5000 M3	Cycle call in connection with point table TAB1.PNT
16 L Z+100 R0 FMAX M6	Retract the tool, change the tool
17 TOOL CALL 3 Z S200	Call tool: tap
18 L Z+50 R0 FMAX	Move tool to clearance height
19 CYCL DEF 206 TAPPING NEW	Cycle definition for tapping
Q200=2 ;SET-UP CLEARANCE	
Q201=-25 ;DEPTH OF THREAD	
Q206=150 ;FEED RATE FOR PLNGNG	
Q211=0 ;DWELL TIME AT DEPTH	
Q203=+0 ;SURFACE COORDINATE	0 must be entered here, effective as defined in point table
Q204=0 ;2ND SET-UP CLEARANCE	0 must be entered here, effective as defined in point table
20 CYCL CALL PAT F5000 M3	Cycle call in connection with point table TAB1.PNT
21 L Z+100 R0 FMAX M2	Retract the tool, end program
22 END PGM 1 MM	

Point table TAB1.PNT

```
TAB1.PNTMM
NRXYZ
0+10+10+0
1+40+30+0
2+90+10+0
3+80+30+0
4+80+65+0
5+90+90+0
6+10+90+0
7+20+55+0
[END]
```



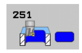
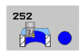
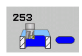
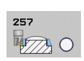
5

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling

5.1 Fundamentals

Overview

The TNC offers 6 cycles for machining pockets, studs and slots:

Cycle	Soft key	Page
251 RECTANGULAR POCKET Roughing/finishing cycle with selection of machining operation and helical plunging		Page 141
252 CIRCULAR POCKET Roughing/finishing cycle with selection of machining operation and helical plunging		Page 146
253 SLOT MILLING Roughing/finishing cycle with selection of machining operation and reciprocal plunging		Page 150
254 CIRCULAR SLOT Roughing/finishing cycle with selection of machining operation and reciprocal plunging		Page 155
256 RECTANGULAR STUD Roughing/finishing cycle with stepover, if multiple passes are required		Page 161
257 CIRCULAR STUD Roughing/finishing cycle with stepover, if multiple passes are required		Page 165

5.2 RECTANGULAR POCKET (Cycle 251, DIN/ISO: G251)

Cycle run

Use Cycle 251 RECTANGULAR POCKET to completely machine rectangular pockets. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, floor finishing, side finishing
- Only roughing
- Only floor finishing and side finishing
- Only floor finishing
- Only side finishing

Roughing

1. The tool plunges into the workpiece at the pocket center and advances to the first plunging depth. Specify the plunging strategy with parameter Q366.
2. The TNC roughs out the pocket from the inside out, taking the overlap factor (parameter Q370) and the finishing allowances (parameters Q368 and Q369) into account.
3. At the end of the roughing operation, the TNC moves the tool tangentially away from the pocket wall, then moves by the set-up clearance above the current infeed depth and returns from there at rapid traverse to the pocket center.
4. This process is repeated until the programmed pocket depth is reached.

Finishing

5. Inasmuch as finishing allowances are defined, the TNC then finishes the pocket walls, in multiple infeeds if so specified. The pocket wall is approached tangentially.
6. Then the TNC finishes the floor of the pocket from the inside out. The pocket floor is approached tangentially.

Please note while programming:

With an inactive tool table you must always plunge vertically (Q366=0) because you cannot define a plunging angle.

Pre-position the tool in the machining plane to the starting position with radius compensation **R0**. Note parameter Q367 (pocket position).

The TNC carries out the cycle in the axes (machining plane) in which you approached the start position. For example, in X and Y if you programmed **CYCL CALL POS X... Y...** or in U and V if you programmed **CYCL CALL POS U... V...**

The TNC automatically pre-positions the tool in the tool axis. Note parameter Q204 (2nd set-up clearance).

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

At the end of the cycle, the TNC returns the tool to the starting position.

At the end of a roughing operation, the TNC positions the tool back to the pocket center at rapid traverse. The tool is above the current pecking depth by the set-up clearance. Enter the set-up clearance so that the tool cannot jam because of chips.

If you mirror Cycle 251 in one axis, the TNC reverses the machining direction defined in the cycle.

Danger of collision!

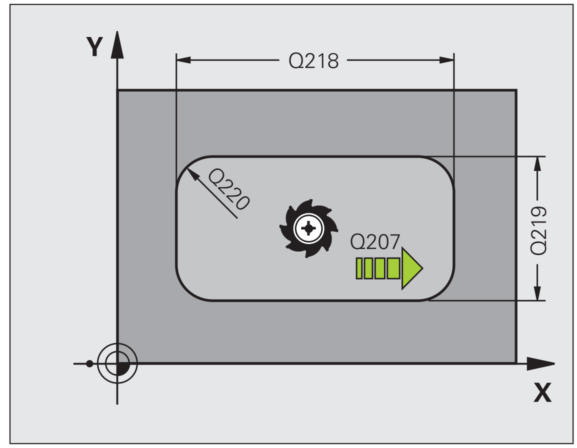
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

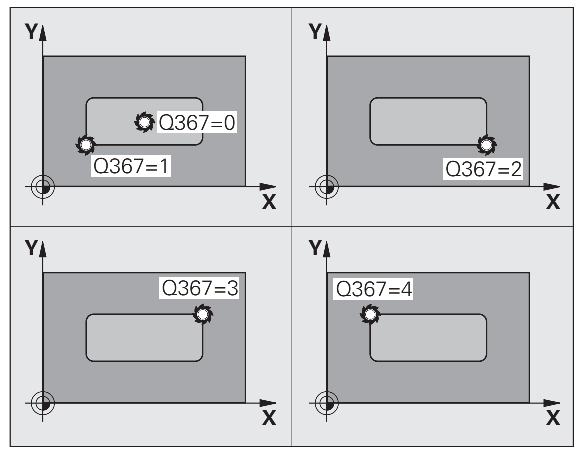
Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

If you call the cycle with machining operation 2 (only finishing), then the TNC positions the tool in the center of the pocket at rapid traverse to the first plunging depth.

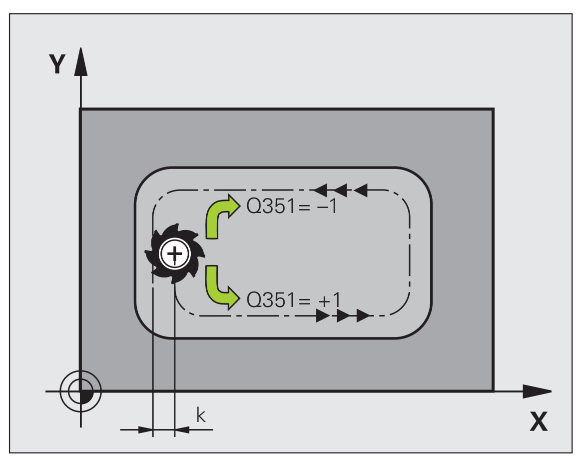
Cycle parameters



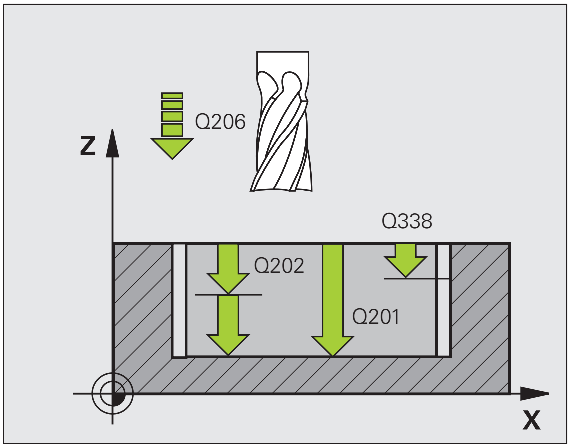
- **`Machining operation (0/1/2)`** Q215: Define the machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only executed if the respective finishing allowance (Q368, Q369) has been defined
- **`1st side length`** Q218 (incremental): Pocket length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- **`2nd side length`** Q219 (incremental): Pocket length, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- **`Corner radius`** Q220: Radius of the pocket corner. If you have entered 0 or a value smaller than the tool radius, the TNC defines the corner radius to be equal to the tool radius. In these cases, the TNC will not display an error message. Input range 0 to 99999.9999
- **`Finishing allowance for side`** Q368 (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999
- **`Angle of rotation`** Q224 (absolute): Angle by which the entire pocket is rotated. The center of rotation is the position at which the tool is located when the cycle is called. Input range -360.0000 to 360.0000

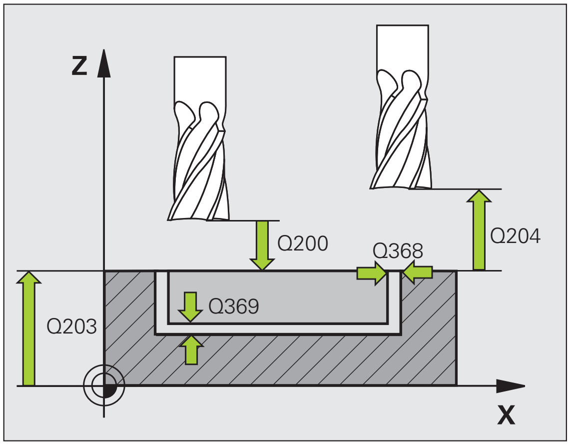


- **`Pocket position`** Q367: Position of the pocket in reference to the position of the tool when the cycle is called:
  0: Tool position = Center of pocket
  1: Tool position = Lower left corner
  2: Tool position = Lower right corner
  3: Tool position = Upper right corner
  4: Tool position = Upper left corner
- **`Feed rate for milling`** Q207: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999; alternatively **`FAUTO`**, **`FU`**, **`FZ`**
- **`Climb or up-cut`** Q351: Type of milling operation with M3:
  +1 = climb milling
  –1 = up-cut milling
  +0 = climb milling; if mirroring is active the TNC stays with climb milling
  Alternatively **`PREDEF`**



- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of pocket. Input range -99999.9999 to 99999.9999
- **Plunging depth** Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999
- **Finishing allowance for floor** Q369 (incremental value): Finishing allowance in the tool axis. Input range 0 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min while moving to depth. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Infeed for finishing** Q338 (incremental): Infeed per cut. Q338=0: Finishing in one infeed. Input range 0 to 99999.9999
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Workpiece surface coordinate** Q203 (absolute): Absolute coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**





- **Path overlap factor** Q370: Q370 x tool radius = stepover factor k. Input range: 0.1 to 1.414; alternatively **PREDEF**
- **Plunging strategy** Q366: Type of plunging strategy:
  - 0 = Vertical plunging. The TNC plunges perpendicularly, regardless of the plunging angle **ANGLE** defined in the tool table.
  - 1 = Helical plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. The TNC will otherwise display an error message.
  - 2 = Reciprocating plunge. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the TNC generates an error message. The reciprocation length depends on the plunging angle. As a minimum value the TNC uses twice the tool diameter.
  - Alternatively **PREDEF**
- **Feed rate for finishing** Q385: Traversing speed of the tool in mm/min during side and floor finishing. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**

Example: NC blocks

```
8 CYCL DEF 251 RECTANGULAR POCKET
    Q215=0     ;MACHINING OPERATION
    Q218=80    ;1ST SIDE LENGTH
    Q219=60    ;2ND SIDE LENGTH
    Q220=5     ;CORNER RADIUS
    Q368=0.2   ;ALLOWANCE FOR SIDE
    Q224=+0    ;ANGLE OF ROTATION
    Q367=0     ;POCKET POSITION
    Q207=500   ;FEED RATE FOR MILLING
    Q351=+1    ;CLIMB OR UP-CUT
    Q201=-20   ;DEPTH
    Q202=5     ;PLUNGING DEPTH
    Q369=0.1   ;ALLOWANCE FOR FLOOR
    Q206=150   ;FEED RATE FOR PLNGNG
    Q338=5     ;INFEED FOR FINISHING
    Q200=2     ;SET-UP CLEARANCE
    Q203=+0    ;SURFACE COORDINATE
    Q204=50    ;2ND SET-UP CLEARANCE
    Q370=1     ;TOOL PATH OVERLAP
    Q366=1     ;PLUNGE
    Q385=500   ;FEED RATE FOR FINISHING
9 CYCL CALL POS X+50 Y+50 Z+0 FMAX M3
```

5.3 CIRCULAR POCKET (Cycle 252, DIN/ISO: G252)

Cycle run

Use Cycle 252 CIRCULAR POCKET to completely machine circular pockets. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, floor finishing, side finishing
- Only roughing
- Only floor finishing and side finishing
- Only floor finishing
- Only side finishing

Roughing

1. The tool plunges into the workpiece at the pocket center and advances to the first plunging depth. Specify the plunging strategy with parameter Q366.
2. The TNC roughs out the pocket from the inside out, taking the overlap factor (parameter Q370) and the finishing allowances (parameters Q368 and Q369) into account.
3. At the end of the roughing operation, the TNC moves the tool tangentially away from the pocket wall, then moves by the set-up clearance above the current infeed depth and returns from there at rapid traverse to the pocket center.
4. This process is repeated until the programmed pocket depth is reached.

Finishing

5. Inasmuch as finishing allowances are defined, the TNC then finishes the pocket walls, in multiple infeeds if so specified. The pocket wall is approached tangentially.
6. Then the TNC finishes the floor of the pocket from the inside out. The pocket floor is approached tangentially.

Please note while programming:

With an inactive tool table you must always plunge vertically (Q366=0) because you cannot define a plunging angle.

Pre-position the tool in the machining plane to the starting position (circle center) with radius compensation **R0**.

The TNC carries out the cycle in the axes (machining plane) in which you approached the start position. For example, in X and Y if you programmed **CYCL CALL POS X... Y...** or in U and V if you programmed **CYCL CALL POS U... V...**

The TNC automatically pre-positions the tool in the tool axis. Note parameter Q204 (2nd set-up clearance).

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

At the end of the cycle, the TNC returns the tool to the starting position.

At the end of a roughing operation, the TNC positions the tool back to the pocket center at rapid traverse. The tool is above the current pecking depth by the set-up clearance. Enter the set-up clearance so that the tool cannot jam because of chips.

If you mirror Cycle 252, the TNC will keep the the machining direction defined in the cycle, i.e. the machining direction will not be mirrored.

Danger of collision!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

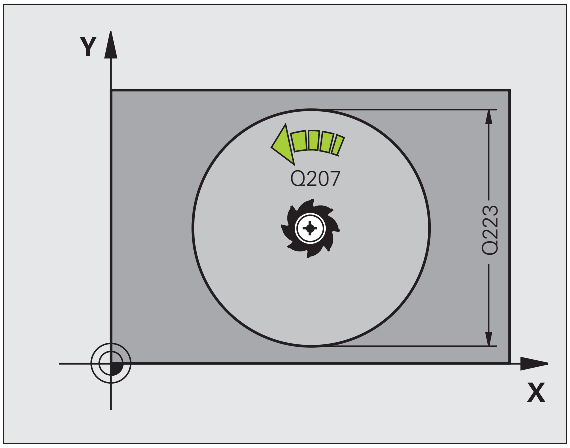
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

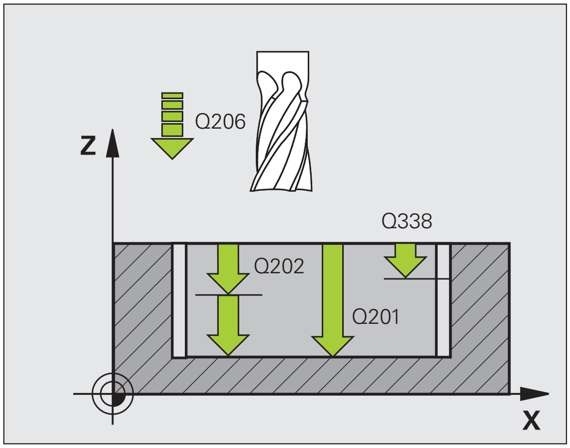
Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

If you call the cycle with machining operation 2 (only finishing), then the TNC positions the tool in the center of the pocket at rapid traverse to the first plunging depth.

Cycle parameters

- **`Machining operation (0/1/2)`** Q215: Define the machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only executed if the respective finishing allowance (Q368, Q369) has been defined
- **`Circle diameter`** Q223: Diameter of the finished pocket. Input range 0 to 99999.9999
- **`Finishing allowance for side`** Q368 (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999
- **`Feed rate for milling`** Q207: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999; alternatively **`FAUTO`**, **`FU`**, **`FZ`**
- **`Climb or up-cut`** Q351: Type of milling operation with M3:
  +1 = climb milling
  –1 = up-cut milling
  +0 = climb milling; if mirroring is active the TNC stays with climb milling
  Alternatively **`PREDEF`**
- **`Depth`** Q201 (incremental): Distance between workpiece surface and bottom of pocket. Input range -99999.9999 to 99999.9999
- **`Plunging depth`** Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999
- **`Finishing allowance for floor`** Q369 (incremental value): Finishing allowance in the tool axis. Input range 0 to 99999.9999
- **`Feed rate for plunging`** Q206: Traversing speed of the tool in mm/min while moving to depth. Input range 0 to 99999.999; alternatively **`FAUTO`**, **`FU`**, **`FZ`**
- **`Infeed for finishing`** Q338 (incremental): Infeed per cut. Q338=0: Finishing in one infeed. Input range 0 to 99999.9999





▶ **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**

▶ **Workpiece surface coordinate** Q203 (absolute): Absolute coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

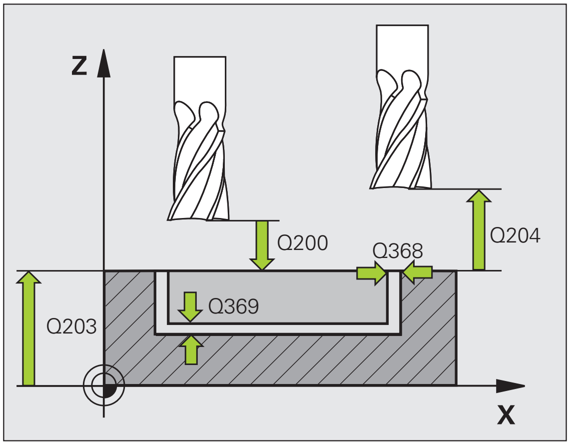
▶ **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**

▶ **Path overlap factor** Q370: Q370 x tool radius = stepover factor k. Input range: 0.1 to 1.414; alternatively **PREDEF**

▶ **Plunging strategy** Q366: Type of plunging strategy:

- 0 = Vertical plunging. The TNC plunges perpendicularly, regardless of the plunging angle **ANGLE** defined in the tool table.
- 1 = Helical plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. The TNC will otherwise display an error message.
- Alternative: **PREDEF**

▶ **Feed rate for finishing** Q385: Traversing speed of the tool in mm/min during side and floor finishing. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**



Example: NC blocks

```
8 CYCL DEF 252 CIRCULAR POCKET
    Q215=0     ;MACHINING OPERATION
    Q223=60    ;CIRCLE DIAMETER
    Q368=0.2   ;ALLOWANCE FOR SIDE
    Q207=500   ;FEED RATE FOR MILLING
    Q351=+1    ;CLIMB OR UP-CUT
    Q201=-20   ;DEPTH
    Q202=5     ;PLUNGING DEPTH
    Q369=0.1   ;ALLOWANCE FOR FLOOR
    Q206=150   ;FEED RATE FOR PLNGNG
    Q338=5     ;INFEED FOR FINISHING
    Q200=2     ;SET-UP CLEARANCE
    Q203=+0    ;SURFACE COORDINATE
    Q204=50    ;2ND SET-UP CLEARANCE
    Q370=1     ;TOOL PATH OVERLAP
    Q366=1     ;PLUNGE
    Q385=500   ;FEED RATE FOR FINISHING
9 CYCL CALL POS X+50 Y+50 Z+0 FMAX M3
```

5.4 SLOT MILLING (Cycle 253, DIN/ISO: G253)

Cycle run

Use Cycle 253 to completely machine a slot. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, floor finishing, side finishing
- Only roughing
- Only floor finishing and side finishing
- Only floor finishing
- Only side finishing

Roughing

1. Starting from the left slot arc center, the tool moves in a reciprocating motion at the plunging angle defined in the tool table to the first infeed depth. Specify the plunging strategy with parameter Q366.
2. The TNC roughs out the slot from the inside out, taking the finishing allowances (parameters Q368 and Q369) into account.
3. This process is repeated until the programmed slot depth is reached.

Finishing

4. Inasmuch as finishing allowances are defined, the TNC then finishes the slot walls, in multiple infeeds if so specified. The slot side is approached tangentially in the right slot arc.
5. Then the TNC finishes the floor of the slot from the inside out. The slot floor is approached tangentially.

Please note while programming:

With an inactive tool table you must always plunge vertically (Q366=0) because you cannot define a plunging angle.

Pre-position the tool in the machining plane to the starting position with radius compensation **R0**. Note parameter Q367 (slot position).

The TNC carries out the cycle in the axes (machining plane) in which you approached the start position. For example, in X and Y if you programmed **CYCL CALL POS X... Y...** or in U and V if you programmed **CYCL CALL POS U... V...**

The TNC automatically pre-positions the tool in the tool axis. Note parameter Q204 (2nd set-up clearance).

At the end of the cycle the TNC merely moves the tool in working plane back to the center of the slot; in the other working plane axis the TNC does not do any positioning. If you define a slot position not equal to 0, then the TNC only positions the tool in the tool axis to the 2nd set-up clearance. Prior to a new cycle call, move the tool back to the starting position or program always absolute traverse motions after the cycle call.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

If the slot width is greater than twice the tool diameter, the TNC roughs the slot correspondingly from the inside out. You can therefore mill any slots with small tools, too.

If you mirror Cycle 253, the TNC will keep the the machining direction defined in the cycle, i.e. the machining direction will not be mirrored.

Danger of collision!

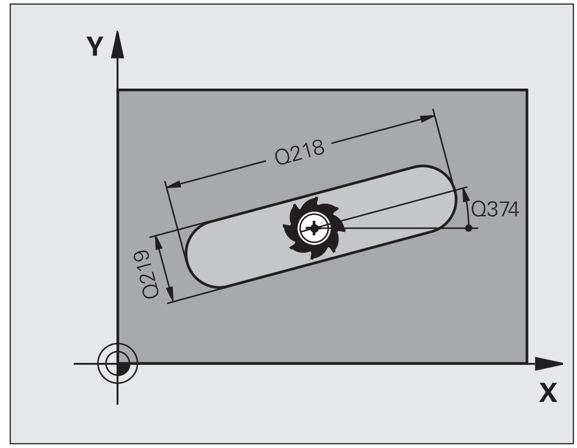
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

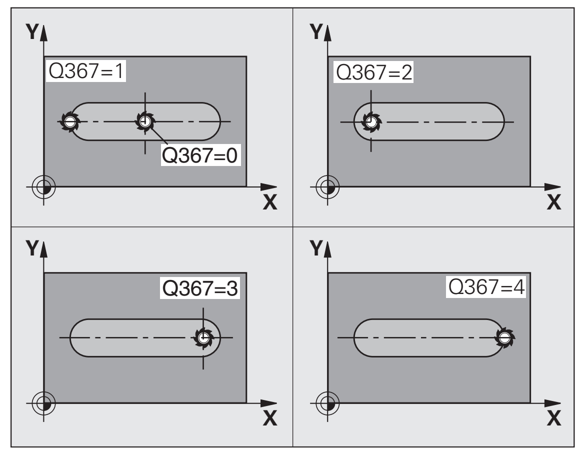
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

If you call the cycle with machining operation 2 (only finishing), then the TNC positions the tool to the first plunging depth at rapid traverse!

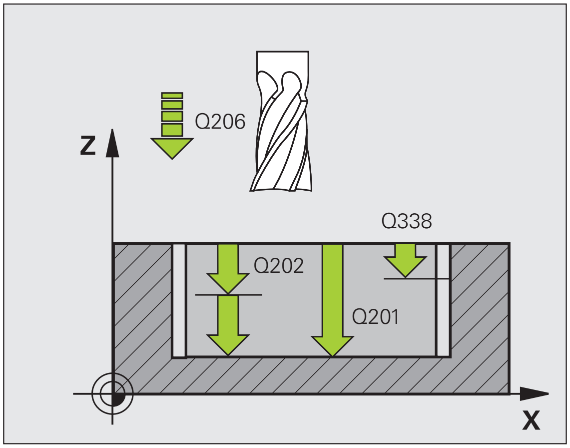
Cycle parameters



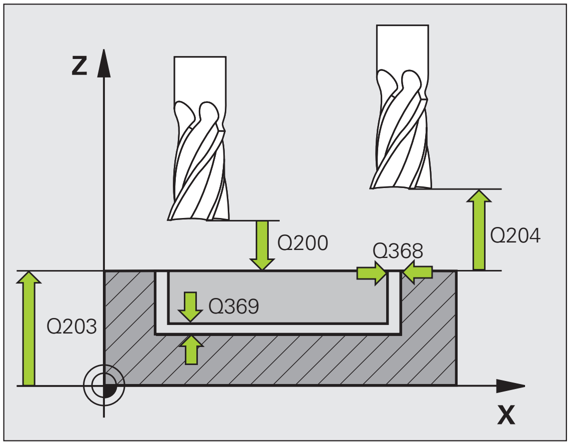


- **`Machining operation (0/1/2)`** Q215: Define the machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only executed if the respective finishing allowance (Q368, Q369) has been defined
- **`Slot length`** Q218 (value parallel to the reference axis of the working plane): Enter the length of the slot. Input range 0 to 99999.9999
- **`Slot width`** Q219 (value parallel to the secondary axis of the working plane): Enter the slot width. If you enter a slot width that equals the tool diameter, the TNC will carry out the roughing process only (slot milling). Maximum slot width for roughing: Twice the tool diameter. Input range 0 to 99999.9999
- **`Finishing allowance for side`** Q368 (incremental): Finishing allowance in the working plane
- **`Angle of rotation`** Q374 (absolute): Angle by which the entire slot is rotated. The center of rotation is the position at which the tool is located when the cycle is called. Input range -360.000 to 360.000
- **`Slot position (0/1/2/3/4)`** Q367: Position of the slot in reference to the position of the tool when the cycle is called:
  0: Tool position = Center of slot
  1: Tool position = Left end of slot
  2: Tool position = Center of left slot circle
  3: Tool position = Center of right slot circle
  4: Tool position = Right end of slot
- **`Feed rate for milling`** Q207: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999; alternatively **`FAUTO`**, **`FU`**, **`FZ`**
- **`Climb or up-cut`** Q351: Type of milling operation with M3:
  +1 = climb milling
  –1 = up-cut milling
  +0 = climb milling; if mirroring is active the TNC stays with climb milling
  Alternatively **`PREDEF`**

- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of slot. Input range -99999.9999 to 99999.9999
- **Plunging depth** Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999
- **Finishing allowance for floor** Q369 (incremental value): Finishing allowance in the tool axis. Input range 0 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min while moving to depth. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Infeed for finishing** Q338 (incremental): Infeed per cut. Q338=0: Finishing in one infeed. Input range 0 to 99999.9999



- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Workpiece surface coordinate** Q203 (absolute): Absolute coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Plunging strategy** Q366: Type of plunging strategy:
  - 0 = Vertical plunging. The TNC plunges perpendicularly, regardless of the plunging angle **ANGLE** defined in the tool table.
  - 1 = Helical plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the TNC generates an error message. Plunge on a helical path only if there is enough space.
  - 2 = Reciprocating plunge. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. The TNC will otherwise display an error message.
  - Alternative: **PREDEF**
- **Feed rate for finishing** Q385: Traversing speed of the tool in mm/min during side and floor finishing. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate reference (0 to 3)** Q439: Selection of what the programmed feed rate refers to
  - 0 = Feed rate refers to the tool path center
  - 1 = Feed rate refers to the tool cutting edge only during side finishing; otherwise it refers to the tool path center
  - 2 = Feed rate refers to the tool cutting edge during side finishing **and** floor finishing; otherwise it refers to the tool path center
  - 3 = Feed rate always refers to the cutting edge



Example: NC blocks

```
8 CYCL DEF 253 SLOT MILLING
    Q215=0     ;MACHINING OPERATION
    Q218=80    ;SLOT LENGTH
    Q219=12    ;SLOT WIDTH
    Q368=0.2   ;ALLOWANCE FOR SIDE
    Q374=+0    ;ANGLE OF ROTATION
    Q367=0     ;SLOT POSITION
    Q207=500   ;FEED RATE FOR MILLING
    Q351=+1    ;CLIMB OR UP-CUT
    Q201=-20   ;DEPTH
    Q202=5     ;PLUNGING DEPTH
    Q369=0.1   ;ALLOWANCE FOR FLOOR
    Q206=150   ;FEED RATE FOR PLNGNG
    Q338=5     ;INFEED FOR FINISHING
    Q200=2     ;SET-UP CLEARANCE
    Q203=+0    ;SURFACE COORDINATE
    Q204=50    ;2ND SET-UP CLEARANCE
    Q366=1     ;PLUNGE
    Q385=500   ;FEED RATE FOR FINISHING
    Q439=0     ;FEED RATE REFERENCE
9 CYCL CALL POS X+50 Y+50 Z+0 FMAX M3
```

5.5 CIRCULAR SLOT (Cycle 254, DIN/ISO: G254)

Cycle run

Use Cycle 254 to completely machine a circular slot. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, floor finishing, side finishing
- Only roughing
- Only floor finishing and side finishing
- Only floor finishing
- Only side finishing

Roughing

1. The tool moves in a reciprocating motion in the slot center at the plunging angle defined in the tool table to the first infeed depth. Specify the plunging strategy with parameter Q366.
2. The TNC roughs out the slot from the inside out, taking the finishing allowances (parameters Q368 and Q369) into account.
3. This process is repeated until the programmed slot depth is reached.

Finishing

4. Inasmuch as finishing allowances are defined, the TNC then finishes the slot walls, in multiple infeeds if so specified. The slot side is approached tangentially.
5. Then the TNC finishes the floor of the slot from the inside out. The slot floor is approached tangentially.

Please note while programming:

With an inactive tool table you must always plunge vertically (Q366=0) because you cannot define a plunging angle.

Pre-position the tool in the machining plane with radius compensation **R0**. Define parameter Q367 **(reference for slot position)** appropriately.

The TNC carries out the cycle in the axes (machining plane) in which you approached the start position. For example, in X and Y if you programmed **CYCL CALL POS X... Y...** or in U and V if you programmed **CYCL CALL POS U... V...**

The TNC automatically pre-positions the tool in the tool axis. Note parameter Q204 (2nd set-up clearance).

At the end of the cycle the TNC merely moves the tool in working plane back to the center of the pitch circle; in the other working plane axis the TNC does not do any positioning. If you define a slot position not equal to 0, then the TNC only positions the tool in the tool axis to the 2nd set-up clearance. Prior to a new cycle call, move the tool back to the starting position or program always absolute traverse motions after the cycle call.

At the end of the cycle the TNC returns the tool to the starting point (center of the pitch circle) in the working plane. Exception: if you define a slot position not equal to 0, then the TNC only positions the tool in the tool axis to the 2nd set-up clearance. In these cases, always program absolute traverse movements after the cycle call.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

If the slot width is greater than twice the tool diameter, the TNC roughs the slot correspondingly from the inside out. You can therefore mill any slots with small tools, too.

The slot position 0 is not allowed if you use Cycle 254 Circular Slot in combination with Cycle 221.

If you mirror Cycle 254, the TNC will keep the the machining direction defined in the cycle, i.e. the machining direction will not be mirrored.

Danger of collision!

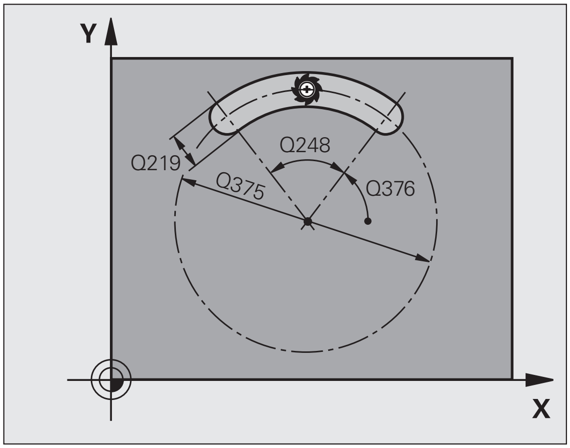
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

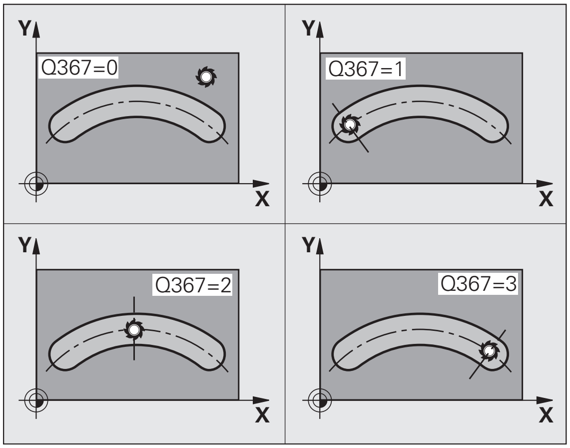
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

If you call the cycle with machining operation 2 (only finishing), then the TNC positions the tool to the first plunging depth at rapid traverse!

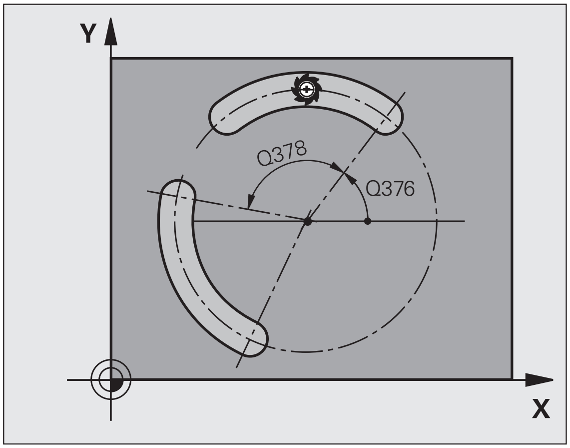
Cycle parameters

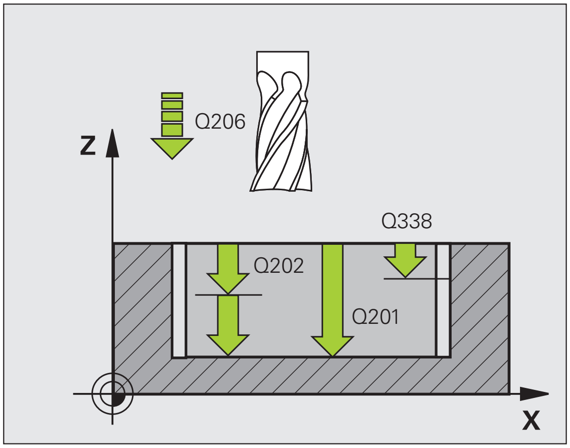


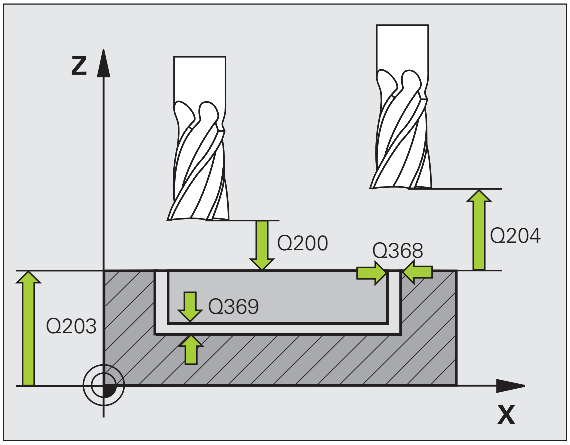


- **Machining operation (0/1/2)** Q215: Define the machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only executed if the respective finishing allowance (Q368, Q369) has been defined
- **Slot width** Q219 (value parallel to the secondary axis of the working plane): Enter the slot width. If you enter a slot width that equals the tool diameter, the TNC will carry out the roughing process only (slot milling). Maximum slot width for roughing: Twice the tool diameter. Input range 0 to 99999.9999
- **Finishing allowance for side** Q368 (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999
- **Pitch circle diameter** Q375: Enter the diameter of the pitch circle. Input range 0 to 99999.9999
- **Reference for slot position (0/1/2/3)** Q367: Position of the slot in reference to the position of the tool when the cycle is called:
  0: The tool position is not taken into account. The slot position is determined from the entered pitch circle center and the starting angle.
  1: Tool position = Center of left slot circle. Starting angle Q376 refers to this position. The entered pitch circle center is not taken into account.
  2: Tool position = Center of center line. Starting angle Q376 refers to this position. The entered pitch circle center is not taken into account.
  3: Tool position = Center of right slot circle. Starting angle Q376 refers to this position. The entered pitch circle center is not taken into account.
- **Center in 1st axis** Q216 (absolute): Center of the pitch circle in the reference axis of the working plane. **Only effective if Q367 = 0**. Input range -99999.9999 to 99999.9999

- **Center in 2nd axis** Q217 (absolute): Center of the pitch circle in the minor axis of the working plane. **Only effective if Q367 = 0**. Input range -99999.9999 to 99999.9999
- **Starting angle** Q376 (absolute): Enter the polar angle of the starting point. Input range -360.000 to 360.000
- **Angular length** Q248 (incremental): Enter the angular length of the slot. Input range 0 to 360.000
- **Stepping angle** Q378 (incremental): Angle by which the entire slot is rotated. The center of rotation is at the center of the pitch circle. Input range -360.000 to 360.000
- **Number of repetitions** Q377: Number of machining operations on a pitch circle. Input range 1 to 99999
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Climb or up-cut** Q351: Type of milling operation with M3:
  +1 = climb milling
  –1 = up-cut milling
  +0 = climb milling; if mirroring is active the TNC stays with climb milling
  Alternatively **PREDEF**
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of slot. Input range -99999.9999 to 99999.9999
- **Plunging depth** Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999
- **Finishing allowance for floor** Q369 (incremental value): Finishing allowance in the tool axis. Input range 0 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min while moving to depth. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Infeed for finishing** Q338 (incremental): Infeed per cut. Q338=0: Finishing in one infeed. Input range 0 to 99999.9999







- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Workpiece surface coordinate** Q203 (absolute): Absolute coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Plunging strategy** Q366: Type of plunging strategy:
  - 0 = vertical plunging. The TNC plunges perpendicularly, regardless of the plunging angle **ANGLE** defined in the tool table.
  - 1 = helical plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the TNC generates an error message. Plunge on a helical path only if there is enough space.
  - 2 = Reciprocating plunge. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the TNC generates an error message. The TNC can only plunge reciprocally once the traversing length on the circular arc is at least three times the tool diameter.
  - Alternative: **PREDEF**
- **Feed rate for finishing** Q385: Traversing speed of the tool in mm/min during side and floor finishing. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate reference (0 to 3)** Q439: Selection of what the programmed feed rate refers to
  - 0 = Feed rate refers to the tool path center
  - 1 = Feed rate refers to the tool cutting edge only during side finishing; otherwise it refers to the tool path center
  - 2 = Feed rate refers to the tool cutting edge during floor finishing **and** side finishing; otherwise it refers to the tool path center
  - 3 = Feed rate always refers to the cutting edge

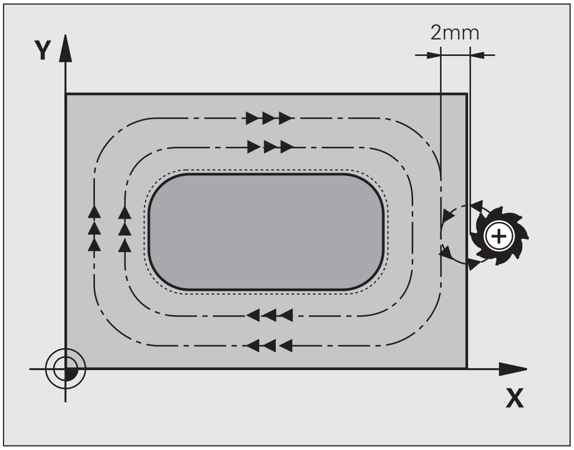
Example: NC blocks

```
8 CYCL DEF 254 CIRCULAR SLOT
     Q215=0     ;MACHINING OPERATION
     Q219=12    ;SLOT WIDTH
     Q368=0.2   ;ALLOWANCE FOR SIDE
     Q375=80    ;PITCH CIRCLE DIA.
     Q367=0     ;REF. SLOT POSITION
     Q216=+50   ;CENTER IN 1ST AXIS
     Q217=+50   ;CENTER IN 2ND AXIS
     Q376=+45   ;STARTING ANGLE
     Q248=90    ;ANGULAR LENGTH
     Q378=0     ;STEPPING ANGLE
     Q377=1     ;NUMBER OF OPERATIONS
     Q207=500   ;FEED RATE FOR MILLING
     Q351=+1    ;CLIMB OR UP-CUT
     Q201=-20   ;DEPTH
     Q202=5     ;PLUNGING DEPTH
     Q369=0.1   ;ALLOWANCE FOR FLOOR
     Q206=150   ;FEED RATE FOR PLNGNG
     Q338=5     ;INFEED FOR FINISHING
     Q200=2     ;SET-UP CLEARANCE
     Q203=+0    ;SURFACE COORDINATE
     Q204=50    ;2ND SET-UP CLEARANCE
     Q366=1     ;PLUNGE
     Q385=500   ;FEED RATE FOR FINISHING
     Q439=0     ;FEED RATE REFERENCE
9 CYCL CALL POS X+50 Y+50 Z+0 FMAX M3
```

5.6 RECTANGULAR STUD (Cycle 256, DIN/ISO: G256)

Cycle run

Use Cycle 256 to machine a rectangular stud. If a dimension of the workpiece blank is greater than the maximum possible stepover, then the TNC performs multiple stepovers until the finished dimension has been machined.



1 The tool moves from the cycle starting position (stud center) to the starting position for stud machining. Specify the starting position with parameter Q437. The standard setting (**Q437=0**) lies 2 mm to the right next to the stud blank
2 If the tool is at the 2nd set-up clearance, it moves at rapid traverse **FMAX** to the set-up clearance, and from there it advances to the first plunging depth at the feed rate for plunging.
3 The tool then moves tangentially to the stud contour and machines one revolution.
4 If the finished dimension cannot be machined with one revolution, the TNC performs a stepover with the current factor, and machines another revolution. The TNC takes the dimensions of the workpiece blank, the finished dimension, and the permitted stepover into account. This process is repeated until the defined finished dimension has been reached. If you have set the starting point on a corner (Q437 not equal to 0), the TNC mills on a spiral path from the starting point inward until the finished dimension has been reached.
5 If more than one plunging movement is required, the tool departs the contour on a tangential path and returns to the starting point of stud machining.
6 The TNC then plunges the tool to the next plunging depth, and machines the stud at this depth.
7 This process is repeated until the programmed stud depth is reached.
8 At the end of the cycle, the TNC merely positions the tool in the tool axis at the clearance height defined in the cycle. This means that the end position differs from the starting position.

Please note while programming:

Pre-position the tool in the machining plane to the starting position with radius compensation **R0**. Note parameter Q367 (stud position).

The TNC automatically pre-positions the tool in the tool axis. Note parameter Q204 (2nd set-up clearance).

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Danger of collision!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

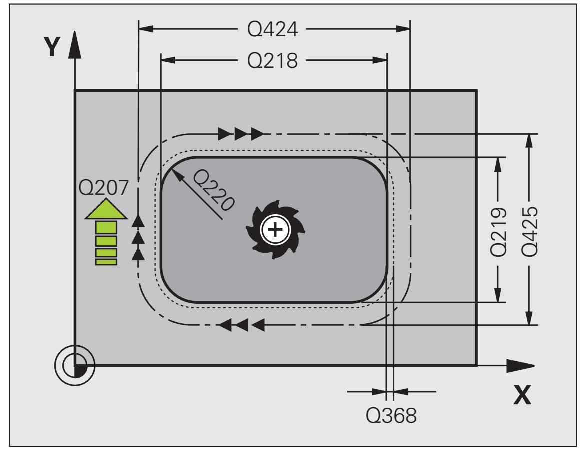
Leave enough room next to the stud for the approach motion. Minimum: Tool diameter plus 2 mm if you are working with standard approach radius and approach angle.

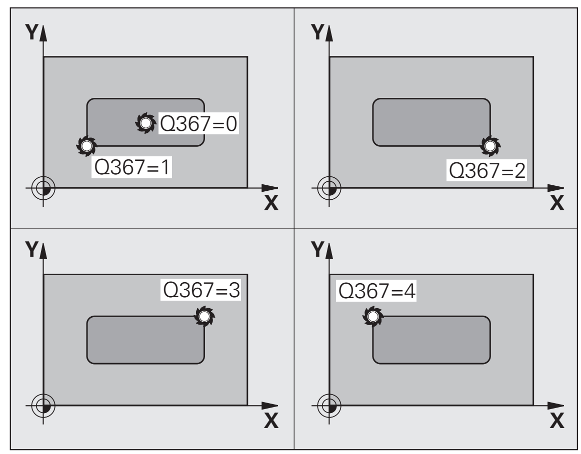
At the end, the TNC positions the tool back to the set-up clearance, or to the 2nd set-up clearance if one was programmed. This means that the end position of the tool after the cycle differs from the starting position.

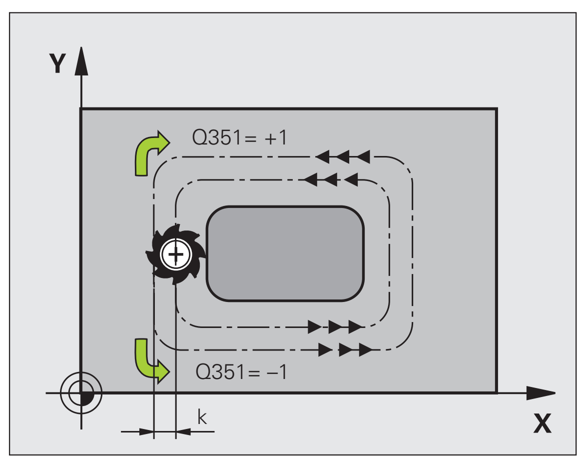
Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

Cycle parameters

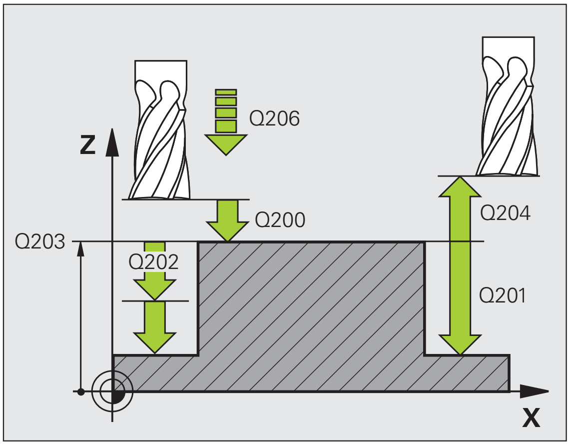
- **1st side length** Q218: Stud length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- **Workpiece blank side length 1** Q424: Length of the stud blank, parallel to the reference axis of the working plane. Enter **Workpiece blank side length 1** greater than **1st side length.** The TNC performs multiple stepovers if the difference between blank dimension 1 and finished dimension 1 is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The TNC always calculates a constant stepover. Input range 0 to 99999.9999
- **2nd side length** Q219: Stud length, parallel to the minor axis of the working plane. Enter **Workpiece blank side length 2** greater than **2nd side length.** The TNC performs multiple stepovers if the difference between blank dimension 2 and finished dimension 2 is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The TNC always calculates a constant stepover. Input range 0 to 99999.9999
- **Workpiece blank side length 2** Q425: Length of the stud blank, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- **Corner radius** Q220: Radius of the stud corner. Input range 0 to 99999.9999
- **Finishing allowance for side** Q368 (incremental): Finishing allowance in the working plane, is left over after machining. Input range 0 to 99999.9999
- **Angle of rotation** Q224 (absolute): Angle by which the entire stud is rotated. The center of rotation is the position at which the tool is located when the cycle is called. Input range -360.000 to 360.000
- **Stud position** Q367: Position of the stud in reference to the position of the tool when the cycle is called:
  0: Tool position = Center of stud
  1: Tool position = Lower left corner
  2: Tool position = Lower right corner
  3: Tool position = Upper right corner
  4: Tool position = Upper left corner







- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Climb or up-cut** Q351: Type of milling operation with M3:
  +1 = climb milling
  –1 = up-cut milling
  Alternatively **PREDEF**
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of stud. Input range -99999.9999 to 99999.9999
- **Plunging depth** Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min while moving to depth. Input range 0 to 99999.999; alternatively **FMAX**, **FAUTO**, **FU**, **FZ**
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Workpiece surface coordinate** Q203 (absolute): Absolute coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Path overlap factor** Q370: Q370 x tool radius = stepover factor k. Input range: 0.1 to 1.414; alternatively **PREDEF**
- **Approach position (0...4)** Q437: Specify the tool approach strategy:
  0: From the right of the stud (default setting)
  1: Lower left corner
  2: Lower right corner
  3: Upper right corner
  4: Upper left corner
  If approach marks should be appear on the stud surface during approach with the setting Q437=0, then choose another approach position



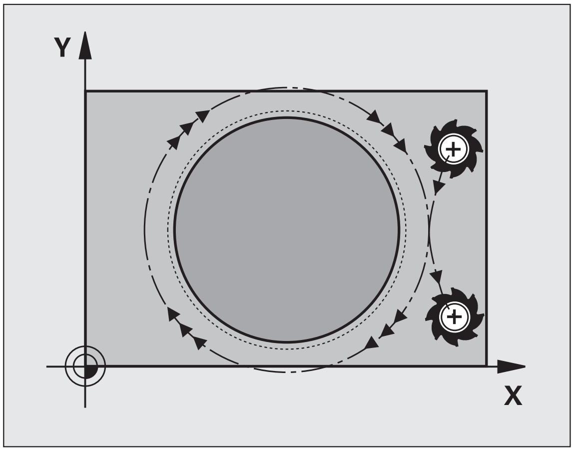
Example: NC blocks

```
8 CYCL DEF 256 RECTANGULAR STUD
    Q218=60    ;1ST SIDE LENGTH
    Q424=74    ;WORKPC. BLANK SIDE 1
    Q219=40    ;2ND SIDE LENGTH
    Q425=60    ;WORKPC. BLANK SIDE 2
    Q220=5     ;CORNER RADIUS
    Q368=0.2   ;ALLOWANCE FOR SIDE
    Q224=+0    ;ANGLE OF ROTATION
    Q367=0     ;STUD POSITION
    Q207=500   ;FEED RATE FOR MILLING
    Q351=+1    ;CLIMB OR UP-CUT
    Q201=-20   ;DEPTH
    Q202=5     ;PLUNGING DEPTH
    Q206=150   ;FEED RATE FOR PLNGNG
    Q200=2     ;SET-UP CLEARANCE
    Q203=+0    ;SURFACE COORDINATE
    Q204=50    ;2ND SET-UP CLEARANCE
    Q370=1     ;TOOL PATH OVERLAP
    Q437=0     ;APPROACH POSITION
9 CYCL CALL POS X+50 Y+50 Z+0 FMAX M3
```

5.7 CIRCULAR STUD (Cycle 257, DIN/ISO: G257)

Cycle run

Use Cycle 257 to machine a circular stud. If the diameter of the workpiece blank is greater than the maximum possible stepover, then the TNC performs a spiral infeed until the finished diameter has been machined.



1. The tool moves from the cycle starting position (stud center) to the starting position for stud machining. With the polar angle you specify the starting position with respect to the stud center using parameter Q376.
2. If the tool is at the 2nd set-up clearance, it moves at rapid traverse **FMAX** to the set-up clearance, and from there it advances to the first plunging depth at the feed rate for plunging.
3. The tool then moves tangentially on a helical path to the stud contour and machines one revolution.
4. If the finished diameter cannot be machined with one revolution, the TNC performs helical infeed movements until the finished diameter is reached. The TNC takes the dimensions of the workpiece blank diameter, the finished diameter, and the permitted stepover into account.
5. The TNC retracts the tool on a helical path from the contour.
6. If more than one plunging movement is required, the tool repeats the plunging movement at the point next to the departure movement.
7. This process is repeated until the programmed stud depth is reached.
8. At the end of the cycle, the TNC merely positions the tool in the tool axis at the clearance height defined in the cycle. This means that the end position differs from the starting position.

Please note while programming:

Pre-position the tool in the machining plane to the starting position (stud center) with radius compensation **R0**.

The TNC automatically pre-positions the tool in the tool axis. Note parameter Q204 (2nd set-up clearance).

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

At the end of the cycle, the TNC only returns the tool to the starting position in the tool axis, but not in the machining plane.

Danger of collision!

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

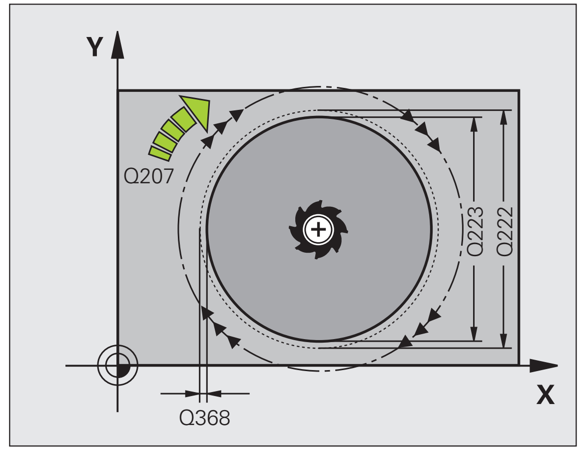
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

Leave enough room next to the stud for the approach motion. Minimum: Tool diameter plus 2 mm if you are working with standard approach radius and approach angle.

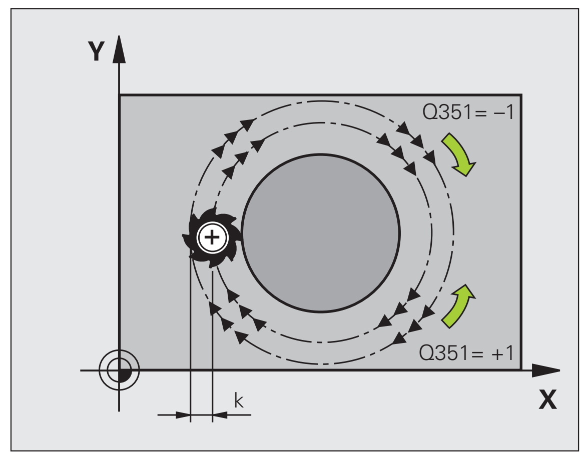
At the end, the TNC positions the tool back to the set-up clearance, or to the 2nd set-up clearance if one was programmed. This means that the end position of the tool after the cycle differs from the starting position.

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

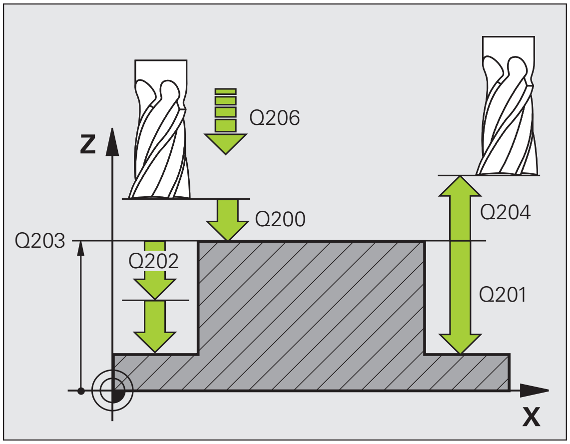
Cycle parameters



- **`Finished part diameter`** Q223: Diameter of the completely machined stud. Input range 0 to 99999.9999
- **`Workpiece blank diameter`** Q222: Diameter of the workpiece blank. Enter the workpiece blank diameter greater than the finished diameter. The TNC performs multiple stepovers if the difference between the workpiece blank diameter and finished diameter is greater than the permitted stepover (tool radius multiplied by path overlap **`Q370`**). The TNC always calculates a constant stepover. Input range 0 to 99999.9999
- **`Finishing allowance for side`** Q368 (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999
- **`Feed rate for milling`** Q207: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999; alternatively **`FAUTO`**, **`FU`**, **`FZ`**
- **`Climb or up-cut`** Q351: Type of milling operation with M3:
  +1 = climb milling
  –1 = up-cut milling
  Alternatively **`PREDEF`**



- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of stud. Input range -99999.9999 to 99999.9999
- **Plunging depth** Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min while moving to depth. Input range 0 to 99999.999; alternatively **FMAX**, **FAUTO**, **FU**, **FZ**
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Workpiece surface coordinate** Q203 (absolute): Absolute coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Path overlap factor** Q370: Q370 x tool radius = stepover factor k. Input range: 0.1 to 1.414; alternatively **PREDEF**
- **Starting angle** Q376: Polar angle relative to the stud center from which the tool approaches the stud. Input range –1 to 359°. The value –1 defines that for repeated downfeeds, the starting angle may vary for each depth, so that the shortest possible paths can be realized. A value between 0 and 359 defines an explicit starting angle that is maintained for every downfeed

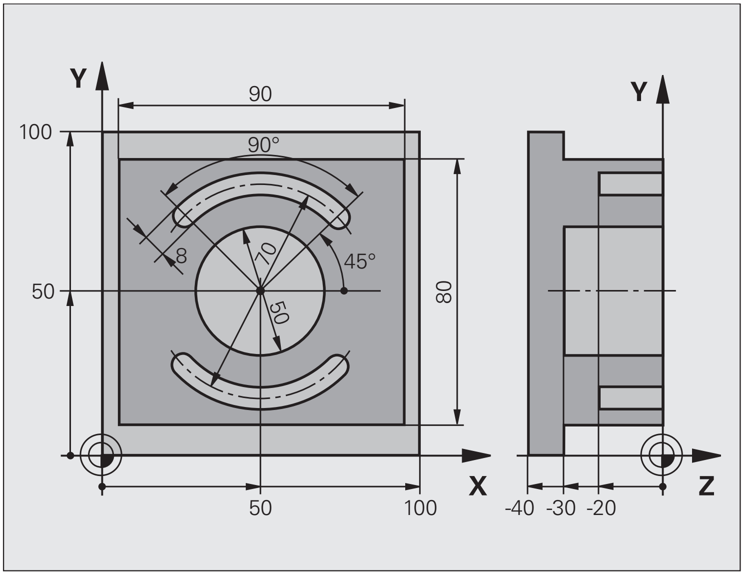


Example: NC blocks

```
8 CYCL DEF 257 CIRCULAR STUD
    Q223=60   ;FINISHED PART DIA.
    Q222=60   ;WORKPIECE BLANK DIA.
    Q368=0.2  ;ALLOWANCE FOR SIDE
    Q207=500  ;FEED RATE FOR MILLING
    Q351=+1   ;CLIMB OR UP-CUT
    Q201=-20  ;DEPTH
    Q202=5    ;PLUNGING DEPTH
    Q206=150  ;FEED RATE FOR PLNGNG
    Q200=2    ;SET-UP CLEARANCE
    Q203=+0   ;SURFACE COORDINATE
    Q204=50   ;2ND SET-UP CLEARANCE
    Q370=1    ;TOOL PATH OVERLAP
    Q376=0    ;STARTING ANGLE
9 CYCL CALL POS X+50 Y+50 Z+0 FMAX M3
```

5.8 Programming examples

Example: Milling pockets, studs and slots



0 BEGIN PGM C210 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL DEF 1 L+0 R+6	Define the tool for roughing/finishing
4 TOOL DEF 2 L+0 R+3	Define slotting mill
5 TOOL CALL 1 Z S3500	Call the tool for roughing/finishing
6 L Z+250 R0 FMAX	Retract the tool
7 CYCL DEF 256 RECTANGULAR STUD	Define cycle for machining the contour outside
Q218=90 ;1ST SIDE LENGTH	
Q424=100 ;WORKPC. BLANK SIDE 1	
Q219=80 ;2ND SIDE LENGTH	
Q425=100 ;WORKPC. BLANK SIDE 2	
Q220=0 ;CORNER RADIUS	
Q368=0 ;ALLOWANCE FOR SIDE	
Q224=0 ;ANGLE OF ROTATION	
Q367=0 ;STUD POSITION	
Q207=250 ;FEED RATE FOR MILLING	
Q351=+1 ;CLIMB OR UP-CUT	

Q201=-30 ;DEPTH	
Q202=5 ;PLUNGING DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q370=1 ;TOOL PATH OVERLAP	
Q437=1 ;APPROACH POSITION	
8 CYCL CALL POS X+50 Y+50 Z+0 M3	Call cycle for machining the contour outside
9 CYCL DEF 252 CIRCULAR POCKET	Define CIRCULAR POCKET MILLING cycle
Q215=0 ;MACHINING OPERATION	
Q223=50 ;CIRCLE DIAMETER	
Q368=0.2 ;ALLOWANCE FOR SIDE	
Q207=500 ;FEED RATE FOR MILLING	
Q351=+1 ;CLIMB OR UP-CUT	
Q201=-30 ;DEPTH	
Q202=5 ;PLUNGING DEPTH	
Q369=0.1 ;ALLOWANCE FOR FLOOR	
Q206=150 ;FEED RATE FOR PLNGNG	
Q338=5 ;INFEED FOR FINISHING	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=50 ;2ND SET-UP CLEARANCE	
Q370=1 ;TOOL PATH OVERLAP	
Q366=1 ;PLUNGE	
Q385=750 ;FEED RATE FOR FINISHING	
10 CYCL CALL POS X+50 Y+50 Z+0 FMAX	Call CIRCULAR POCKET MILLING cycle
11 L Z+250 R0 FMAX M6	Tool change
12 TOOL CALL 2 Z S5000	Call tool: slotting mill
13 CYCL DEF 254 CIRCULAR SLOT	Define SLOT cycle
Q215=0 ;MACHINING OPERATION	
Q219=8 ;SLOT WIDTH	
Q368=0.2 ;ALLOWANCE FOR SIDE	
Q375=70 ;PITCH CIRCLE DIA.	
Q367=0 ;REF. SLOT POSITION	No pre-positioning in X/Y required
Q216=+50 ;CENTER IN 1ST AXIS	
Q217=+50 ;CENTER IN 2ND AXIS	
Q376=+45 ;STARTING ANGLE	

Q248=90 ;ANGULAR LENGTH	
Q378=180 ;STEPPING ANGLE	Starting point for second slot
Q377=2 ;NUMBER OF OPERATIONS	
Q207=500 ;FEED RATE FOR MILLING	
Q351=+1 ;CLIMB OR UP-CUT	
Q201=-20 ;DEPTH	
Q202=5 ;PLUNGING DEPTH	
Q369=0.1 ;ALLOWANCE FOR FLOOR	
Q206=150 ;FEED RATE FOR PLNGNG	
Q338=5 ;INFEED FOR FINISHING	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=50 ;2ND SET-UP CLEARANCE	
Q366=1 ;PLUNGE	
Q439=0 ;FEED RATE REFERENCE	
14 CYCL CALL FMAX M3	Call SLOT cycle
15 L Z+250 R0 FMAX M2	Retract the tool, end program
16 END PGM C210 MM	

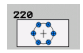
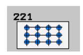
6

Fixed Cycles: Pattern Definitions

6.1 Fundamentals

Overview

The TNC provides two cycles for machining point patterns directly:

Cycle	Soft key	Page
220 POLAR PATTERN		Page 175
221 CARTESIAN PATTERN		Page 178

You can combine Cycle 220 and Cycle 221 with the following fixed cycles:

> If you have to machine irregular point patterns, use **CYCL CALL PAT** (see "Point tables" on page 67) to develop point tables.
>
> More regular point patterns are available with the **PATTERN DEF** function (see "PATTERN DEF pattern definition" on page 59).

Cycle 200	DRILLING
Cycle 201	REAMING
Cycle 202	BORING
Cycle 203	UNIVERSAL DRILLING
Cycle 204	BACK BORING
Cycle 205	UNIVERSAL PECKING
Cycle 206	TAPPING NEW with a floating tap holder
Cycle 207	RIGID TAPPING without a floating tap holder NEW
Cycle 208	BORE MILLING
Cycle 209	TAPPING WITH CHIP BREAKING
Cycle 240	CENTERING
Cycle 251	RECTANGULAR POCKET
Cycle 252	CIRCULAR POCKET MILLING
Cycle 253	SLOT MILLING
Cycle 254	CIRCULAR SLOT (can only be combined with Cycle 221)
Cycle 256	RECTANGULAR STUD
Cycle 257	CIRCULAR STUD
Cycle 262	THREAD MILLING
Cycle 263	THREAD MILLING/COUNTERSINKING
Cycle 264	THREAD DRILLING/MILLING
Cycle 265	HELICAL THREAD DRILLING/MILLING
Cycle 267	OUTSIDE THREAD MILLING

6.2 POLAR PATTERN (Cycle 220, DIN/ISO: G220)

Cycle run

1 The TNC moves the tool at rapid traverse from its current position to the starting point for the first machining operation.

Sequence:

- Move to the 2nd set-up clearance (spindle axis)
- Approach the starting point in the machining plane
- Move to the set-up clearance above the workpiece surface (spindle axis)

2 From this position the TNC executes the last defined fixed cycle.

3 The tool then approaches on a straight line or circular arc the starting point for the next machining operation. The tool stops at the set-up clearance (or the 2nd set-up clearance).

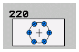
4 This process (1 to 3) is repeated until all machining operations have been executed.

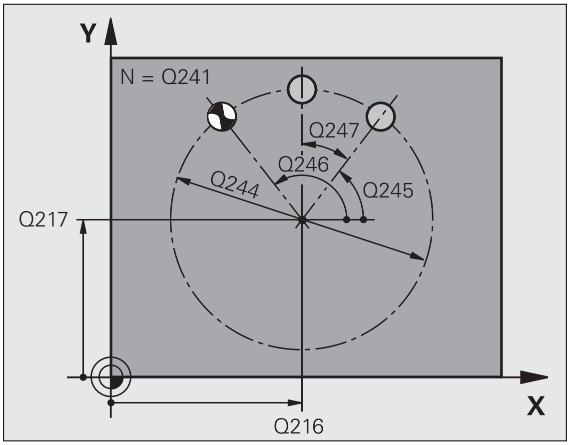
Please note while programming:

Cycle 220 is DEF active, which means that Cycle 220 automatically calls the last defined fixed cycle.

If you combine Cycle 220 with one of the fixed cycles 200 to 209 and 251 to 267, the set-up clearance, workpiece surface and 2nd set-up clearance from Cycle 220 will be effective for the selected fixed cycle.

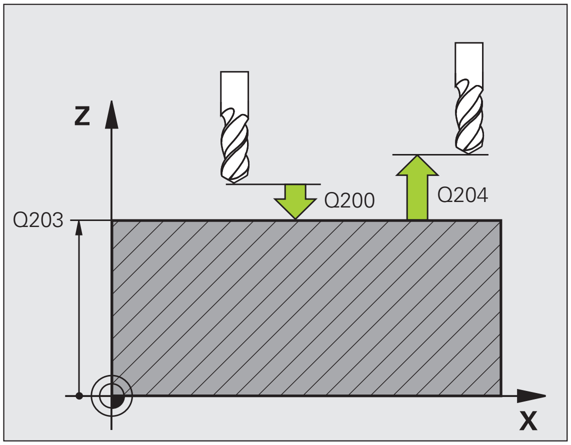
Cycle parameters





- **Center in 1st axis** Q216 (absolute): Center of the pitch circle in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q217 (absolute): Center of the pitch circle in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Pitch circle diameter** Q244: Diameter of the pitch circle. Input range 0 to 99999.9999
- **Starting angle** Q245 (absolute): Angle between the reference axis of the working plane and the starting point for the first machining operation on the pitch circle. Input range -360.000 to 360.000
- **Stopping angle** Q246 (absolute): Angle between the reference axis of the working plane and the starting point for the last machining operation on the pitch circle (does not apply to full circles). Do not enter the same value for the stopping angle and starting angle. If you enter the stopping angle greater than the starting angle, machining will be carried out counterclockwise; otherwise, machining will be clockwise. Input range -360.000 to 360.000
- **Stepping angle** Q247 (incremental): Angle between two machining operations on a pitch circle. If you enter an angle step of 0, the TNC will calculate the angle step from the starting and stopping angles and the number of pattern repetitions. If you enter a value other than 0, the TNC will not take the stopping angle into account. The sign for the angle step determines the working direction (negative = clockwise). Input range -360.000 to 360.000
- **Number of repetitions** Q241: Number of machining operations on a pitch circle. Input range 1 to 99999

- **`Set-up clearance`** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **`PREDEF`**
- **`Coordinate of workpiece surface`** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **`2nd set-up clearance`** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **`PREDEF`**
- **`Move to clearance height`** Q301: Definition of how the tool is to move between machining processes:
  0: Move to the set-up clearance between machining operations
  1: Move to the 2nd set-up clearance between machining operations
  Alternatively **`PREDEF`**
- **`Type of traverse? Line=0/Arc=1`** Q365: Definition of the path function with which the tool moves between machining operations:
  0: Move in a straight line between machining operations
  1: Move in a circular arc on the pitch circle diameter between machining operations

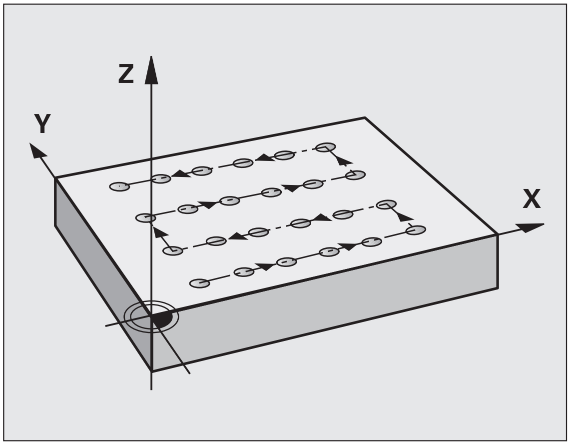


Example: NC blocks

```
53 CYCL DEF 220 POLAR PATTERN
    Q216=+50  ;CENTER IN 1ST AXIS
    Q217=+50  ;CENTER IN 2ND AXIS
    Q244=80   ;PITCH CIRCLE DIA.
    Q245=+0   ;STARTING ANGLE
    Q246=+360 ;STOPPING ANGLE
    Q247=+0   ;STEPPING ANGLE
    Q241=8    ;NUMBER OF OPERATIONS
    Q200=2    ;SET-UP CLEARANCE
    Q203=+30  ;SURFACE COORDINATE
    Q204=50   ;2ND SET-UP CLEARANCE
    Q301=1    ;MOVE TO CLEARANCE
    Q365=0    ;TYPE OF TRAVERSE
```

6.3 CARTESIAN PATTERN (Cycle 221, DIN/ISO: G221)

Cycle run



1 The TNC automatically moves the tool from its current position to the starting point for the first machining operation.

Sequence:

- Move to the 2nd set-up clearance (spindle axis)
- Approach the starting point in the machining plane
- Move to the set-up clearance above the workpiece surface (spindle axis)

2 From this position the TNC executes the last defined fixed cycle.

3 The tool then approaches the starting point for the next machining operation in the positive reference axis direction at the set-up clearance (or the 2nd set-up clearance).

4 This process (1 to 3) is repeated until all machining operations on the first line have been executed. The tool is located above the last point on the first line.

5 The tool subsequently moves to the last point on the second line where it executes the machining operation.

6 From this position the tool approaches the starting point for the next machining operation in the negative reference axis direction.

7 This process (6) is repeated until all machining operations in the second line have been executed.

8 The tool then moves to the starting point of the next line.

9 All subsequent lines are machined in a reciprocating movement.

Please note while programming:

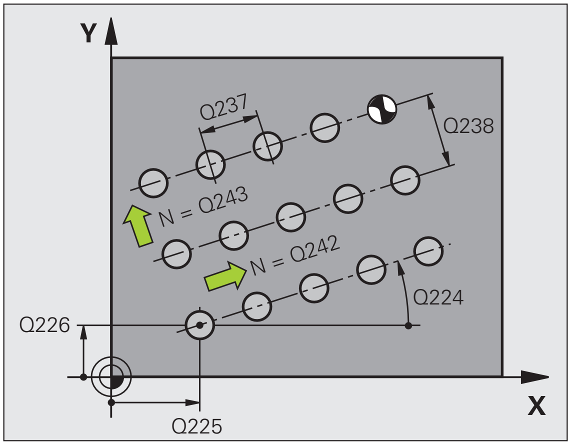
Cycle 221 is DEF active, which means that Cycle 221 automatically calls the last defined fixed cycle.

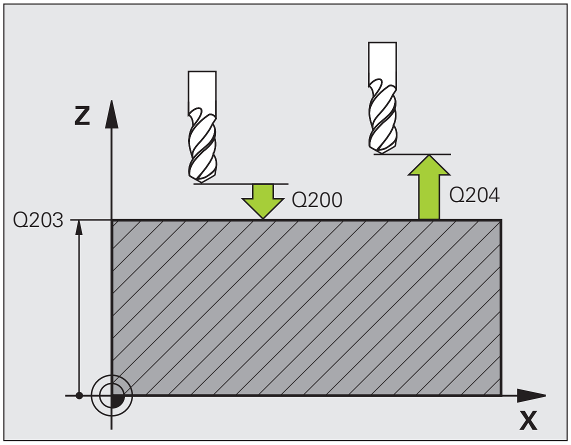
If you combine Cycle 221 with one of the fixed cycles 200 to 209 and 251 to 267, the set-up clearance, workpiece surface, 2nd set-up clearance and rotational position that you defined in Cycle 221 will be effective for the selected fixed cycle.

The slot position 0 is not allowed if you use Cycle 254 Circular Slot in combination with Cycle 221.

Cycle parameters

- **Starting point 1st axis** Q225 (absolute): Coordinate of the starting point in the reference axis of the working plane
- **Starting point 2nd axis** Q226 (absolute): Coordinate of the starting point in the minor axis of the working plane
- **Spacing in 1st axis** Q237 (incremental): Spacing between each point on a line
- **Spacing in 2nd axis** Q238 (incremental): Spacing between each line
- **Number of columns** Q242: Number of machining operations on a line
- **Number of lines** Q243: Number of lines
- **Rotational position** Q224 (absolute): Angle by which the entire pattern is rotated. The center of rotation lies in the starting point
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface; alternatively **PREDEF**
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur; alternatively **PREDEF**
- **Move to clearance height** Q301: Definition of how the tool is to move between machining processes:
  0: Move to the set-up clearance between machining operations
  1: Move to the 2nd set-up clearance between machining operations
  Alternatively **PREDEF**



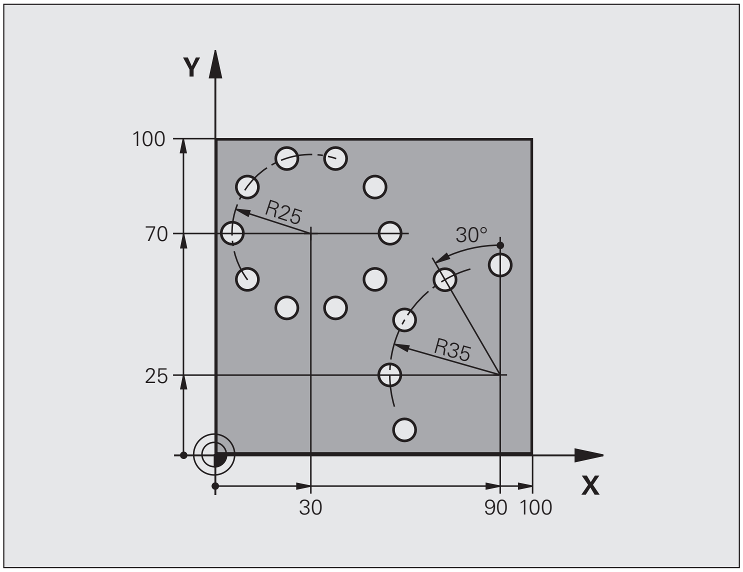


Example: NC blocks

```
54 CYCL DEF 221 CARTESIAN PATTERN
    Q225=+15   ;STARTING POINT 1ST AXIS
    Q226=+15   ;STARTING POINT 2ND AXIS
    Q237=+10   ;SPACING IN 1ST AXIS
    Q238=+8    ;SPACING IN 2ND AXIS
    Q242=6     ;NUMBER OF COLUMNS
    Q243=4     ;NUMBER OF LINES
    Q224=+15   ;ANGLE OF ROTATION
    Q200=2     ;SET-UP CLEARANCE
    Q203=+30   ;SURFACE COORDINATE
    Q204=50    ;2ND SET-UP CLEARANCE
    Q301=1     ;MOVE TO CLEARANCE
```

6.4 Programming examples

Example: Polar hole patterns



0 BEGIN PGM PATTERN MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL DEF 1 L+0 R+3	Tool definition
4 TOOL CALL 1 Z S3500	Tool call
5 L Z+250 R0 FMAX M3	Retract the tool
6 CYCL DEF 200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-15 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=4 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME	
Q203=+0 ;SURFACE COORDINATE	
Q204=0 ;2ND SET-UP CLEARANCE	
Q211=0.25 ;DWELL TIME AT DEPTH	
Q395=0.25 ;DEPTH REFERENCE	

7 CYCL DEF 220 POLAR PATTERN	Define cycle for polar pattern 1, CYCL 200 is called automatically; Q200, Q203 and Q204 are effective as defined in Cycle 220.
Q216=+30 ;CENTER IN 1ST AXIS	
Q217=+70 ;CENTER IN 2ND AXIS	
Q244=50 ;PITCH CIRCLE DIA.	
Q245=+0 ;STARTING ANGLE	
Q246=+360 ;STOPPING ANGLE	
Q247=+0 ;STEPPING ANGLE	
Q241=10 ;NUMBER OF REPETITIONS	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=100 ;2ND SET-UP CLEARANCE	
Q301=1 ;MOVE TO CLEARANCE	
Q365=0 ;TYPE OF TRAVERSE	
8 CYCL DEF 220 POLAR PATTERN	Define cycle for polar pattern 2, CYCL 200 is called automatically; Q200, Q203 and Q204 are effective as defined in Cycle 220.
Q216=+90 ;CENTER IN 1ST AXIS	
Q217=+25 ;CENTER IN 2ND AXIS	
Q244=70 ;PITCH CIRCLE DIA.	
Q245=+90 ;STARTING ANGLE	
Q246=+360 ;STOPPING ANGLE	
Q247=30 ;STEPPING ANGLE	
Q241=5 ;NUMBER OF REPETITIONS	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=100 ;2ND SET-UP CLEARANCE	
Q301=1 ;MOVE TO CLEARANCE	
Q365=0 ;TYPE OF TRAVERSE	
9 L Z+250 R0 FMAX M2	Retract the tool, end program
10 END PGM PATTERN MM	

7

Fixed Cycles: Contour Pocket, Contour Trains

7.1 SL cycles

Fundamentals

SL cycles enable you to form complex contours by combining up to 12 subcontours (pockets or islands). You define the individual subcontours in subprograms. The TNC calculates the total contour from the subcontours (subprogram numbers) that you enter in Cycle 14 CONTOUR.

> The memory capacity for programming an SL cycle (all contour subprograms) is limited. The number of possible contour elements depends on the type of contour (inside or outside contour) and the number of subcontours. You can program up to 8192 contour elements.
>
> SL cycles conduct comprehensive and complex internal calculations as well as the resulting machining operations. For safety reasons, always run a graphical program test before machining! This is a simple way of finding out whether the TNC-calculated program will provide the desired results.

Characteristics of the subprograms

- Coordinate transformations are allowed. If they are programmed within the subcontour they are also effective in the following subprograms, but they need not be reset after the cycle call.
- The TNC ignores feed rates F and miscellaneous functions M.
- The TNC recognizes a pocket if the tool path lies inside the contour, for example if you machine the contour clockwise with radius compensation RR.
- The TNC recognizes an island if the tool path lies outside the contour, for example if you machine the contour clockwise with radius compensation RL.
- The subprograms must not contain spindle axis coordinates.
- The working plane is defined in the first coordinate block of the subprogram. The secondary axes U,V,W are permitted in useful combinations. Always define both axes of the machining plane in the first block.
- If you use Q parameters, then only perform the calculations and assignments within the affected contour subprograms.
- If an open contour is defined in the subprogram, the TNC uses a straight line from the end point to the starting point to close the contour.

Example: Program structure: Machining with SL cycles

```
0 BEGIN PGM SL2 MM
...
12 CYCL DEF 14 CONTOUR GEOMETRY ...
13 CYCL DEF 20 CONTOUR DATA ...
...
16 CYCL DEF 21 PILOT DRILLING ...
17 CYCL CALL
...
18 CYCL DEF 22 ROUGH-OUT ...
19 CYCL CALL
...
22 CYCL DEF 23 FLOOR FINISHING ...
23 CYCL CALL
...
26 CYCL DEF 24 SIDE FINISHING ...
27 CYCL CALL
...
50 L Z+250 R0 FMAX M2
51 LBL 1
...
55 LBL 0
56 LBL 2
...
60 LBL 0
...
99 END PGM SL2 MM
```

Characteristics of the fixed cycles

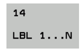
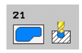
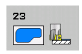
- The TNC automatically positions the tool to the set-up clearance before a cycle.
- Each level of infeed depth is milled without interruptions since the cutter traverses around islands instead of over them.
- In order to avoid leaving dwell marks, the TNC inserts a globally definable rounding radius at non-tangential inside corners. The rounding radius, which is entered in Cycle 20, affects the tool center point path, meaning that it would increase a rounding defined by the tool radius (applies to rough-out and side finishing).
- The contour is approached in a tangential arc for side finishing.
- For floor finishing, the tool again approaches the workpiece on a tangential arc (for tool axis Z, for example, the arc is in the Z/X plane).
- The contour is machined throughout in either climb or up-cut milling.

With bit 4 in MP7420 you can determine where the tool is positioned at the end of Cycles 21 to 24.

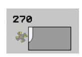
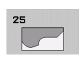
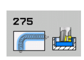
- **bit 4 = 0:**
  At the end of the cycle, the TNC at first positions the tool in the tool axis at the clearance height (**Q7**) defined in the cycle, and then to the position in the working plane at which the tool was located when the cycle was called.
- **Bit 4 = 1:**
  At the end of the cycle, the TNC always positions the tool in the tool axis at the clearance height (**Q7**) defined in the cycle. Ensure that no collisions can occur during the following positioning movements!

The machining data (such as milling depth, finishing allowance and set-up clearance) are entered as CONTOUR DATA in Cycle 20.

Overview

Cycle	Soft key	Page
14 CONTOUR GEOMETRY (essential)		Page 187
20 CONTOUR DATA (compulsory)		Page 192
21 PILOT DRILLING (optional)		Page 194
22 ROUGH-OUT (essential)		Page 196
23 FLOOR FINISHING (optional)		Page 200
24 SIDE FINISHING (optional)		Page 202

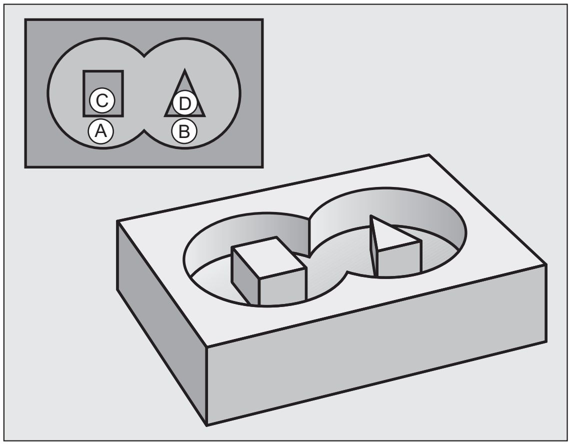
Enhanced cycles:

Cycle	Soft key	Page
270 CONTOUR TRAIN DATA		Page 204
25 CONTOUR TRAIN		Page 206
275 TROCHOIDAL SLOT		Page 210
276 THREE-D CONT. TRAIN		Page 215

7.2 CONTOUR (Cycle 14, DIN/ISO: G37)

Please note while programming:

All subprograms that are superimposed to define the contour are listed in Cycle 14 CONTOUR GEOMETRY.



Before programming, note the following:

Cycle 14 is DEF active which means that it becomes effective as soon as it is defined in the part program.

You can list up to 12 subprograms (subcontours) in Cycle 14.

Cycle parameters

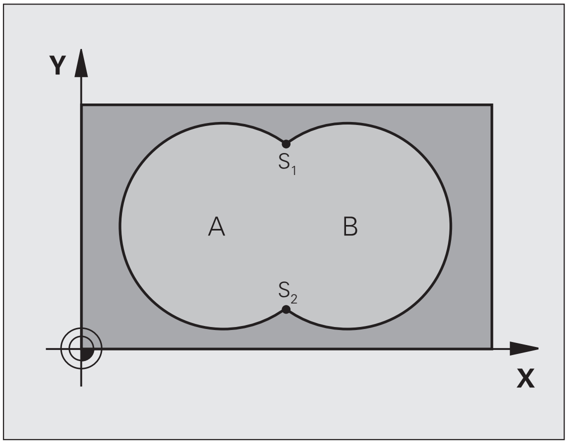
14
LBL 1...N

- **Label numbers for the contour**: Enter all label numbers for the individual subprograms that are to be superimposed to define the contour. Confirm every label number with the ENT key. When you have entered all numbers, conclude entry with the END key. Entry of up to 12 subprogram numbers 1 to 254.

7.3 Overlapping contours

Fundamentals

Pockets and islands can be overlapped to form a new contour. You can thus enlarge the area of a pocket by another pocket or reduce it by an island.



Example: NC blocks

```
12 CYCL DEF 14.0 CONTOUR GEOMETRY
13 CYCL DEF 14.1 CONTOUR LABEL 1/2/3/4
```

Subprograms: overlapping pockets

The subsequent programming examples are contour subprograms that are called by Cycle 14 CONTOUR GEOMETRY in a main program.

Pockets A and B overlap.

The TNC calculates the points of intersection S_1 and S_2. They do not have to be programmed.

The pockets are programmed as full circles.

Subprogram 1: Pocket A

```
51 LBL 1
52 L X+10 Y+50 RR
53 CC X+35 Y+50
54 C X+10 Y+50 DR-
55 LBL 0
```

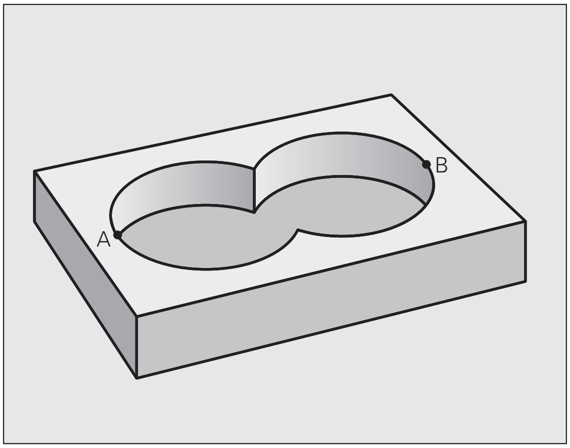
Subprogram 2: Pocket B

```
56 LBL 2
57 L X+90 Y+50 RR
58 CC X+65 Y+50
59 C X+90 Y+50 DR-
60 LBL 0
```

Area of inclusion

Both areas A and B are to be machined, including the overlapping area:

- The surfaces A and B must be pockets.
- The first pocket (in Cycle 14) must start outside the second pocket.



Surface A:

```
51 LBL 1
52 L X+10 Y+50 RR
53 CC X+35 Y+50
54 C X+10 Y+50 DR-
55 LBL 0
```

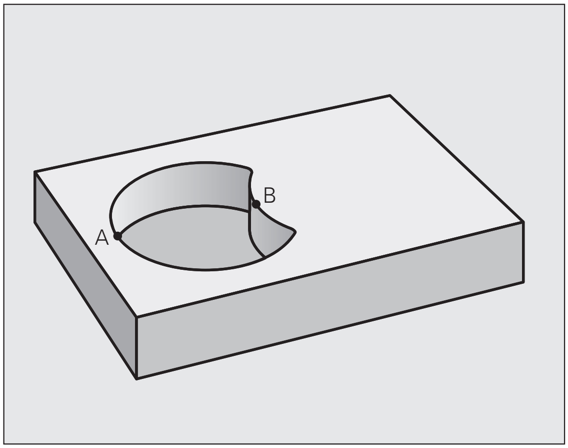
Surface B:

```
56 LBL 2
57 L X+90 Y+50 RR
58 CC X+65 Y+50
59 C X+90 Y+50 DR-
60 LBL 0
```

Area of exclusion

Area A is to be machined without the portion overlapped by B:

- Surface A must be a pocket and B an island.
- A must start outside of B.
- B must start inside of A.



Surface A:

```
51 LBL 1
52 L X+10 Y+50 RR
53 CC X+35 Y+50
54 C X+10 Y+50 DR-
55 LBL 0
```

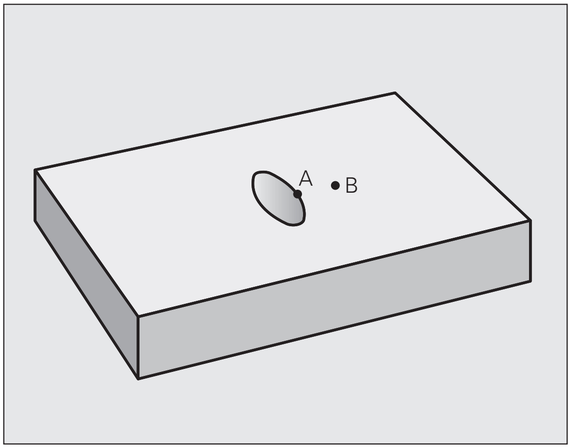
Surface B:

```
56 LBL 2
57 L X+40 Y+50 RL
58 CC X+65 Y+50
59 C X+40 Y+50 DR-
60 LBL 0
```

Area of intersection

Only the area where A and B overlap is to be machined. (The areas covered by A or B alone are to be left unmachined.)

- A and B must be pockets.
- A must start inside of B.



Surface A:

```
51 LBL 1
52 L X+60 Y+50 RR
53 CC X+35 Y+50
54 C X+60 Y+50 DR-
55 LBL 0
```

Surface B:

```
56 LBL 2
57 L X+90 Y+50 RR
58 CC X+65 Y+50
59 C X+90 Y+50 DR-
60 LBL 0
```

7.4 CONTOUR DATA (Cycle 20, DIN/ISO: G120)

Please note while programming:

Machining data for the subprograms describing the subcontours are entered in Cycle 20.

Cycle 20 is DEF active, which means that it becomes effective as soon as it is defined in the part program.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the TNC performs the cycle at the depth 0.

The machining data entered in Cycle 20 are valid for Cycles 21 to 24.

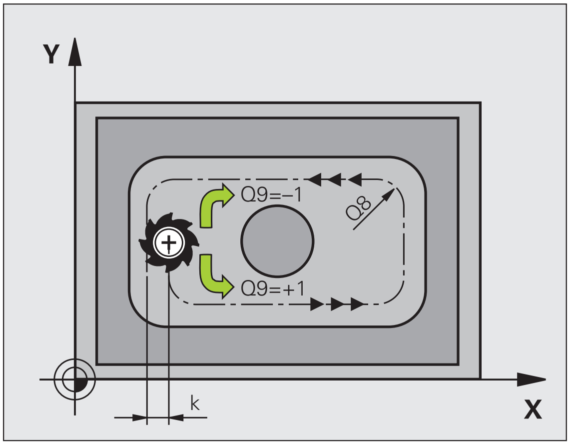
If you are using the SL cycles in Q parameter programs, the cycle parameters Q1 to Q20 cannot be used as program parameters.

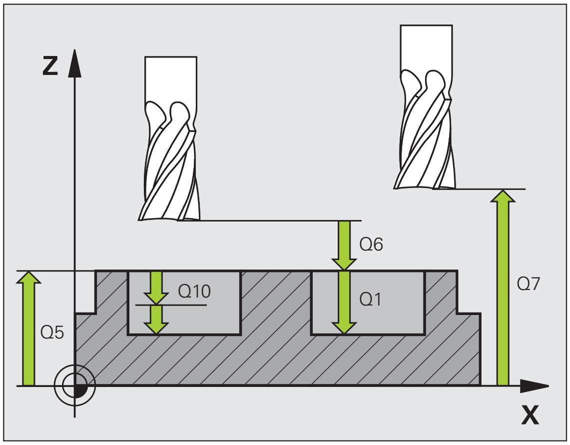
Cycle parameters

20 CONTOUR DATA

- **Milling depth** Q1 (incremental): Distance between workpiece surface and bottom of pocket. Input range -99999.9999 to 99999.9999
- **Path overlap** factor Q2: Q2 x tool radius = stepover factor k. Input range -0.0001 to 1.9999.
- **Finishing allowance for side** Q3 (incremental): Finishing allowance in the working plane. Input range -99999.9999 to 99999.9999
- **Finishing allowance for floor** Q4 (incremental): Finishing allowance in the tool axis. Input range -99999.9999 to 99999.9999
- **Workpiece surface coordinate** Q5 (absolute): Absolute coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q6 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Clearance height** Q7 (absolute): Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). Input range -99999.9999 to 99999.9999; alternatively **PREDEF**
- **Inside corner radius** Q8: Inside "corner" rounding radius; entered value is referenced to the path of the tool center and is used to calculate smoother traverse motions between the contour elements. **Q8 is not a radius that is inserted as a separate contour element between programmed elements!** Input range 0 to 99999.9999
- **Direction of rotation?** Q9: Machining direction for pockets
  - Q9 = –1 up-cut milling for pocket and island
  - Q9 = +1 climb milling for pocket and island
  - Alternative: **PREDEF**

You can check the machining parameters during a program interruption and overwrite them if required.





Example: NC blocks

```
57 CYCL DEF 20 CONTOUR DATA
     Q1=-20      ;MILLING DEPTH
     Q2=1        ;TOOL PATH OVERLAP
     Q3=+0.2     ;ALLOWANCE FOR SIDE
     Q4=+0.1     ;ALLOWANCE FOR FLOOR
     Q5=+30      ;SURFACE COORDINATE
     Q6=2        ;SET-UP CLEARANCE
     Q7=+80      ;CLEARANCE HEIGHT
     Q8=0.5      ;ROUNDING RADIUS
     Q9=+1       ;DIRECTION OF ROTATION
```

7.5 PILOT DRILLING (Cycle 21, DIN/ISO: G121)

Cycle run

1. The tool drills from the current position to the first plunging depth at the programmed feed rate **F**.
2. Then the tool retracts at rapid traverse **FMAX** to the starting position and advances again to the first plunging depth minus the advanced stop distance t.
3. The advanced stop distance is automatically calculated by the control:
   - At a total hole depth up to 30 mm: t = 0.6 mm
   - At a total hole depth exceeding 30 mm: t = hole depth / 50
   - Maximum advanced stop distance: 7 mm
4. The tool then advances with another infeed at the programmed feed rate F.
5. The TNC repeats this process (1 to 4) until the programmed depth is reached
6. After a dwell time at the hole bottom, the tool is returned to the starting position at rapid traverse **FMAX** for chip breaking.

Insert

Cycle 21 is for PILOT DRILLING of the cutter infeed points. It accounts for the allowance for side and the allowance for floor as well as the radius of the rough-out tool. The cutter infeed points also serve as starting points for roughing.

Please note while programming:

Before programming, note the following:

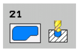
When calculating the infeed points, the TNC does not account for the delta value **DR** programmed in a **TOOL CALL** block.

In narrow areas, the TNC may not be able to carry out pilot drilling with a tool that is larger than the rough-out tool.

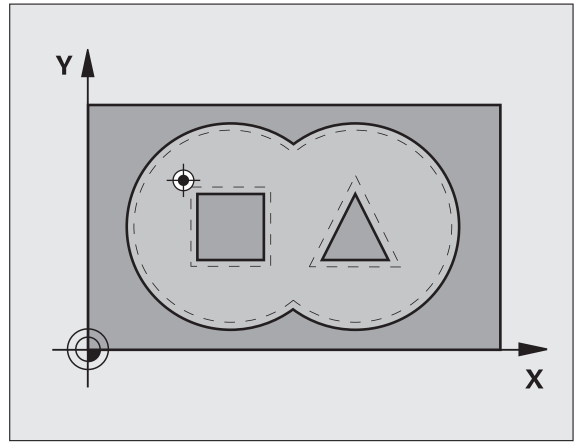
Danger of collision!

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

Cycle parameters



- **`Plunging depth`** Q10 (incremental): Dimension by which the tool drills in each infeed (negative sign for negative working direction). Input range -99999.9999 to 99999.9999
- **`Feed rate for plunging`** Q11: Traversing speed of the tool in mm/min during drilling. Input range 0 to 99999.9999; alternatively **`FAUTO`**, **`FU`**, **`FZ`**
- **`Rough-out tool number/name`** Q13 or QS13: Number or name of rough-out tool. Input range 0 to 32767.9 if a number is entered; maximum 32 characters if a name is entered.

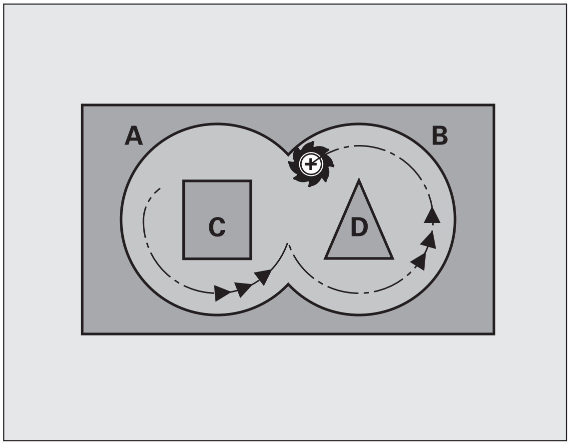


Example: NC blocks

```
58 CYCL DEF 21 PILOT DRILLING
    Q10=+5     ;PLUNGING DEPTH
    Q11=100    ;FEED RATE FOR PLNGNG
    Q13=1      ;ROUGH-OUT TOOL
```

7.6 ROUGH-OUT (Cycle 22, DIN/ISO: G122)

Cycle run



1. The TNC positions the tool over the cutter infeed point, taking the allowance for side into account.
2. In the first plunging depth, the tool mills the contour from the inside outward at the milling feed rate Q12.
3. The island contours (here: C/D) are cleared out with an approach toward the pocket contour (here: A/B)
4. In the next step the TNC moves the tool to the next plunging depth and repeats the roughing procedure until the programmed depth is reached.
5. Finally, the TNC moves the tool to the clearance height, and, if defined, returns it in the machining plane to the position at which the cycle was called (depends on MP7420, bit 4)

Please note while programming:

This cycle requires a center-cut end mill (ISO 1641) or pilot drilling with Cycle 21.

You define the plunging behavior of Cycle 22 with parameter Q19 and with the tool table in the **ANGLE** and **LCUTS** columns:

- If Q19=0 is defined, the TNC always plunges perpendicularly, even if a plunge angle (**ANGLE**) is defined for the active tool.
- If you define the **ANGLE**=90°, the TNC plunges perpendicularly. The reciprocation feed rate Q19 is used as plunging feed rate.
- If the reciprocation feed rate Q19 is defined in Cycle 22 and **ANGLE** is defined between 0.1 and 89.999 in the tool table, the TNC plunges helically at the defined **ANGLE**.
- If the reciprocation feed rate is defined in Cycle 22 and no **ANGLE** is in the tool table, the TNC displays an error message.
- If geometrical conditions do not allow helical plunging (slot geometry), the TNC tries a reciprocating plunge. The reciprocation length is calculated from **LCUTS** and **ANGLE** (reciprocation length = **LCUTS** / tan **ANGLE**).

If you clear out an acute inside corner and use an overlap factor greater than 1, some material might be left over. Check especially the innermost path in the test run graphic and, if necessary, change the overlap factor slightly. This allows another distribution of cuts, which often provides the desired results.

During fine roughing the TNC does not take a defined wear value **DR** of the coarse roughing tool into account.

Feed rate reduction through parameter **Q401** is an FCL3 function and is not automatically available after a software update (see "Feature content level (upgrade functions)" on page 9).

Danger of collision!

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

If you have set **MP7420, bit 4=1**, then after executing the SL cycle you must program the first traverse motion in the working plane with both coordinate values, e.g. **L X+80 Y+0 R0 FMAX**. After the end of the cycle, **do not use incremental positioning** for the tool in the plane. Rather, always program an absolute position.

Cycle parameters

- **Plunging depth** Q10 (incremental): Infeed per cut. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q11: Traversing speed of the tool in mm/min during plunging. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for roughing** Q12: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Coarse roughing tool** Q18 or QS18: Number or name of the tool with which the TNC has already coarse-roughed the contour. Press the TOOL NAME soft key to switch to name input. The TNC automatically inserts the closing quotation mark when you exit the input field. If there was no coarse roughing, enter "0"; if you enter a number or a name, the TNC will only rough-out the portion that could not be machined with the coarse roughing tool. If the portion that is to be roughed cannot be approached from the side, the TNC will mill in a reciprocating plunge-cut; for this purpose you must enter the tool length **LCUTS** in the tool table TOOL.T and define the maximum plunging **ANGLE** of the tool. The TNC will otherwise generate an error message. If any of the software limit switches is traversed the TNC will display an error message. Input range 0 to 32767.9 if a number is entered; maximum 32 characters if a name is entered.
- **Reciprocation feed rate** Q19: Traversing speed of the tool in mm/min during reciprocating plunge-cut. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for retraction** Q208: Traversing speed of the tool in mm/min when retracting after the machining operation. If you enter Q208 = 0, the TNC retracts the tool at the feed rate Q12. Input range 0 to 99999.9999; alternatively **FMAX**, **FAUTO**, **PREDEF**

Example: NC blocks

```
59 CYCL DEF 22 ROUGH-OUT
    Q10=+5    ;PLUNGING DEPTH
    Q11=100   ;FEED RATE FOR PLNGNG
    Q12=750   ;FEED RATE FOR ROUGHING
    Q18=1     ;COARSE ROUGHING TOOL
    Q19=150   ;RECIPROCATION FEED RATE
    Q208=99999;RETRACTION FEED RATE
    Q401=80   ;FEED RATE REDUCTION
    Q404=0    ;FINE ROUGH STRATEGY
```

- **Feed rate factor in %:** Q401: Percentage factor by which the TNC reduces the machining feed rate **(Q12)** as soon as the tool moves within the material over its entire circumference during roughing. If you use the feed rate reduction, then you can define the feed rate for roughing so large that there are optimum cutting conditions with the path overlap **(Q2)** specified in Cycle 20. The TNC then reduces the feed rate as per your definition at transitions and narrow places, so the machining time should be reduced in total. Input range 0.0001 to 100.0000
- **Fine-roughing strategy** Q404: Specify how the TNC should move the tool during fine roughing when the radius of the fine-roughing tool is larger than half the coarse roughing tool.
  - Q404 = 0
    Between areas that need to be fine-roughed, move the tool along the contour at the current depth
  - Q404 = 1
    Between areas that need to be fine-roughed, retract the tool to set-up clearance and move to the starting point of the next area to be roughed out

7.7 FLOOR FINISHING (Cycle 23, DIN/ISO: G123)

Cycle run

The tool approaches the machining plane smoothly (on a vertically tangential arc) if there is sufficient room. If there is not enough room, the TNC moves the tool to depth vertically. The tool then clears the finishing allowance remaining from rough-out.

Please note while programming:

The TNC automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket.

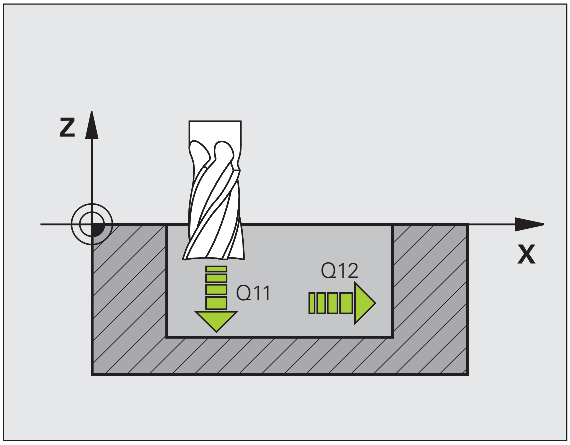
The approaching radius for pre-positioning to the final depth is permanently defined and independent of the plunging angle of the tool.

Danger of collision!

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

Cycle parameters

- **`Feed rate for plunging`** Q11: Traversing speed of the tool during plunging. Input range 0 to 99999.9999; alternatively **`FAUTO`**, **`FU`**, **`FZ`**
- **`Feed rate for roughing`** Q12: Milling feed rate. Input range 0 to 99999.9999; alternatively **`FAUTO`**, **`FU`**, **`FZ`**
- **`Feed rate for retraction`** Q208: Traversing speed of the tool in mm/min when retracting after the machining operation. If you enter Q208 = 0, the TNC retracts the tool at the feed rate Q12. Input range 0 to 99999.9999; alternatively **`FMAX`**, **`FAUTO`**, **`PREDEF`**



Example: NC blocks

```
60 CYCL DEF 23 FLOOR FINISHING
    Q11=100   ;FEED RATE FOR PLNGNG
    Q12=350   ;FEED RATE FOR ROUGHING
    Q208=99999;RETRACTION FEED RATE
```

7.8 SIDE FINISHING (Cycle 24, DIN/ISO: G124)

Cycle run

The individual subcontours are approached and departed on a tangential arc. The TNC finishes each subcontour separately.

Please note while programming:

The sum of allowance for side (Q14) and the radius of the finish mill must be smaller than the sum of allowance for side (Q3, Cycle 20) and the radius of the rough mill.

This calculation also holds if you run Cycle 24 without having roughed out with Cycle 22; in this case, enter "0" for the radius of the rough mill.

You can use Cycle 24 also for contour milling. Then you must:

- define the contour to be milled as a single island (without pocket limit), and
- enter the finishing allowance (Q3) in Cycle 20 to be greater than the sum of the finishing allowance Q14 + radius of the tool being used.

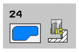
The TNC automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket and the allowance programmed in Cycle 20. The TNC executes the positioning logic to the starting point of the finishing operation as follows: approach the starting point in the working plane, then move to depth in tool axis direction.

The starting point calculated by the TNC also depends on the machining sequence. If you select the finishing cycle with the GOTO key and then start the program, the starting point can be at a different location from where it would be if you execute the program in the defined sequence.

Danger of collision!

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

Cycle parameters

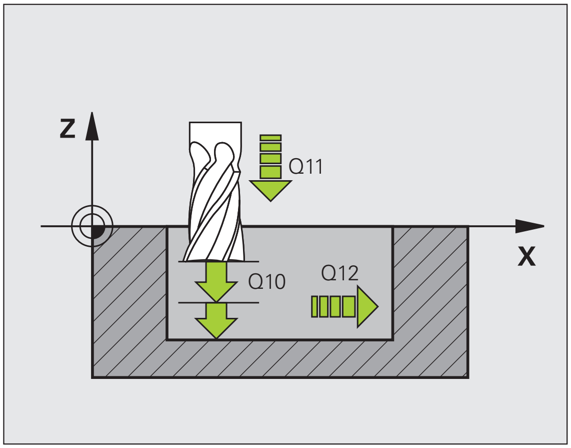


- **Direction of rotation? Clockwise = –1** Q9: Machining direction:
  +1:Counterclockwise
  –1:Clockwise
  Alternative: **PREDEF**
- **Plunging depth** Q10 (incremental): Infeed per cut. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q11: Traversing speed of the tool during plunging. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for roughing** Q12: Milling feed rate. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Finishing allowance for side** Q14 (incremental): Enter the allowed material for several finish-milling operations. If you enter Q14 = 0, the remaining finishing allowance will be cleared. Input range -99999.9999 to 99999.9999
- **Rough-out tool** Q438 or QS438: Number or name of the tool with which the TNC roughed out the contour pocket. Press the TOOL NAME soft key to switch to name input. The TNC automatically inserts the closing quotation mark when you exit the input field.

  The starting point of the circular arc on which the tool approaches the finishing path is on the outermost roughing path from Cycle 22. The TNC calculates the starting point from the sum of the rough-out tool radius and the finishing allowance for side Q3 defined in Cycle 20. Input range -1 to +30000.9 if a number is entered; maximum 32 characters if a name is entered.

  Q438=–1: Roughing is performed with the most recently used tool (standard behavior)

  Q438=0: A roughing tool with the radius 0 is assumed. This allows you to use the finishing allowance Q3 in Cycle 20 to specify the distance from the starting point to the contour.



Example: NC blocks

```
61 CYCL DEF 24 SIDE FINISHING
    Q9=+1       ;DIRECTION OF ROTATION
    Q10=+5      ;PLUNGING DEPTH
    Q11=100     ;FEED RATE FOR PLNGNG
    Q12=350     ;FEED RATE FOR ROUGHING
    Q14=+0      ;ALLOWANCE FOR SIDE
    Q438=+0     ;ROUGH-OUT TOOL
```

7.9 CONTOUR TRAIN DATA (Cycle 270, DIN/ISO: G270)

Please note while programming:

If desired, you can use this cycle to specify various properties of Cycle 25, **CONTOUR TRAIN** and Cycle 276, **3-D CONTOUR TRAIN**.

Before programming, note the following:

Cycle 270 is DEF active, which means that it becomes effective as soon as it is defined in the part program.

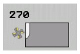
The TNC resets Cycle 270 as soon as you define another SL cycle (with the exception of Cycle 25 and Cycle 276).

If Cycle 270 is used, do not define any radius compensation in the contour subprogram.

Approach and departure properties are always performed identically (symmetrically) by the TNC.

Define Cycle 270 before Cycle 25 or Cycle 276.

Cycle parameters



- **`Type of approach/departure`** Q390: Definition of the type of approach or departure.
  - Q390 = 1:
    Approach the contour tangentially on a circular arc.
  - Q390 = 2:
    Approach the contour tangentially on a straight line.
  - Q390 = 3:
    Approach the contour at a right angle.
- **`Radius compensation (0=R0/1=RL/2=RR)`** Q391: Definition of the radius compensation:
  - Q391 = 0:
    Machine the defined contour without radius compensation.
  - Q391 = 1:
    Machine the defined contour with compensation to the left.
  - Q391 = 2:
    Machine the defined contour with compensation to the right.
- **`Approach/departure radius`** Q392: Only in effect if tangential approach on a circular path was selected. Radius of the approach/departure arc. Input range 0 to 99999.9999
- **`Center angle`** Q393: Only in effect if tangential approach on a circular path was selected. Angular length of the approach arc. Input range 0 to 99999.9999
- **`Distance to auxiliary point`** Q394: Only in effect if tangential approach on a straight line or right-angle approach was selected. Distance to the auxiliary point from which the TNC is to approach the contour. Input range 0 to 99999.9999

Example: NC blocks

```
62 CYCL DEF 270 CONTOUR TRAIN DATA
    Q390=1     ;TYPE OF APPROACH
    Q391=1     ;RADIUS COMPENSATION
    Q392=3     ;RADIUS
    Q393=+45   ;CENTER ANGLE
    Q394=+2    ;DISTANCE
```

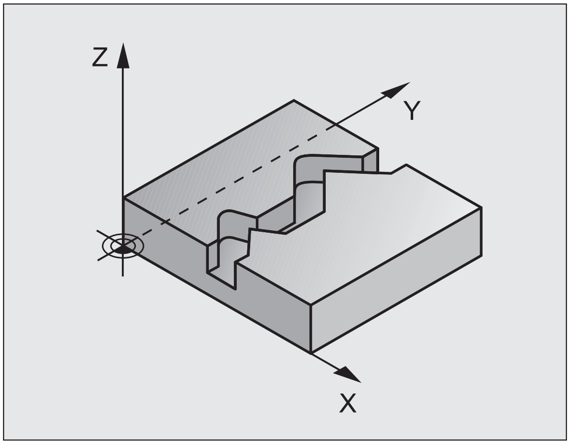
7.10 CONTOUR TRAIN (Cycle 25, DIN/ISO: G125)

Cycle run

In conjunction with Cycle 14 **CONTOUR GEOMETRY**, this cycle facilitates the machining of open and closed contours.

Cycle 25 **CONTOUR TRAIN** offers considerable advantages over machining a contour using positioning blocks:

- The TNC monitors the operation to prevent undercuts and surface blemishes. It is recommended that you run a graphic simulation of the contour before execution.
- If the radius of the selected tool is too large, the inside corners of the contour can be reworked using the feature for **automatic identification of residual material**.
- The contour can be machined throughout by up-cut or by climb milling. The type of milling even remains effective when contours are mirrored in one axis.
- The tool can traverse back and forth for milling in several infeeds (reciprocating machining): This results in faster machining
- Allowance values can be entered in order to perform repeated rough-milling and finish-milling operations.
- Cycle 270 CONTOUR TRAIN DATA provides an easy way to define the behavior of Cycle 25.



Please note while programming:

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

When using Cycle 25 **CONTOUR TRAIN**, you can define only one contour program in Cycle 14 **CONTOUR GEOMETRY**.

The memory capacity for programming an SL cycle is limited. You can program up to 4090 contour elements in one SL cycle.

The TNC does not need Cycle 20 **CONTOUR DATA** in conjunction with Cycle 25.

Do not use any approach or departure blocks **APPR**/**DEP** in the contour subprogram.

Do not perform Q parameter calculations in the contour subprogram.

Use the **CONTOUR TRAIN DATA** cycle to define the machining behavior of Cycle 25 (see "CONTOUR TRAIN DATA (Cycle 270, DIN/ISO: G270)" on page 204).

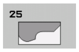
Danger of collision!

To avoid collisions,

- Do not program positions in incremental dimensions immediately after Cycle 25 since they are referenced to the position of the tool at the end of the cycle.
- Move the tool to defined (absolute) positions in all main axes, since the position of the tool at the end of the cycle is not identical to the position of the tool at the start of the cycle.
- If you program **APPR** and **DEP** blocks for contour approach and departure, the TNC monitors whether the execution of these blocks would damage the contour.

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

Cycle parameters



- **Milling depth** Q1 (incremental): Distance between workpiece surface and contour floor. Input range -99999.9999 to 99999.9999
- **Finishing allowance for side** Q3 (incremental): Finishing allowance in the working plane. Input range -99999.9999 to 99999.9999
- **Workpiece surface coordinate** Q5 (absolute): Absolute coordinate of the workpiece surface referenced to the workpiece datum. Input range -99999.9999 to 99999.9999
- **Clearance height** Q7 (absolute): Absolute height at which the tool cannot collide with the workpiece. Position for tool retraction at the end of the cycle. Input range -99999.9999 to 99999.9999; alternatively **PREDEF**
- **Plunging depth** Q10 (incremental): Infeed per cut. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q11: Traversing speed of the tool in the spindle axis. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for milling** Q12: Traversing speed of the tool in the working plane. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Climb/Up-cut? Up-cut = –1** Q15:
  Climb milling: Input value = +1
  Up-cut milling: Input value = –1
  To enable climb milling and up-cut milling alternately in several infeeds:Input value = 0

Example: NC blocks

```
62 CYCL DEF 25 CONTOUR TRAIN
     Q1=-20     ;MILLING DEPTH
     Q3=+0      ;ALLOWANCE FOR SIDE
     Q5=+0      ;SURFACE COORDINATE
     Q7=+50     ;CLEARANCE HEIGHT
     Q10=+5     ;PLUNGING DEPTH
     Q11=100    ;FEED RATE FOR PLNGNG
     Q12=350    ;FEED RATE FOR MILLING
     Q15=-1     ;CLIMB OR UP-CUT
     Q18=0      ;COARSE ROUGHING TOOL
     Q446=0.01 ;RESIDUAL MATERIAL
     Q447=10    ;CONNECTION DISTANCE
     Q448=2     ;PATH EXTENSION
```

- **Coarse roughing tool** Q18 or QS18: Number or name of the tool with which the TNC has already coarse-roughed the contour. Press the TOOL NAME soft key to switch to name input. The TNC automatically inserts the closing quotation mark when you exit the input field. If there was no coarse roughing, enter "0": the TNC will machine the contour as much as is possible withe the active tool; if you enter a number or a name, the TNC will machine only the contour part that could not be machined with the coarse roughing tool. Input range 0 to 32767.9 if a number is entered; maximum 32 characters if a name is entered.
- **Accepted residual material** Q446: Remaining material thickness as of which the TNC should no longer machine the contour. Default value: 0.01 mm Input range 0 to +9.999
- **Maximum connection distance** Q447: Maximum distance between two areas to be fine-roughed, between which the tool is to move at the machining depth along the contour without a lift-off motion. Input range 0 to 999
- **Path extension** Q448: Length by which the tool path is extended at the start and end of the contour. The TNC always extends the tool path parallel to the contour. The approach and departure behavior for fine-roughing must be defined in Cycle 270. Input range 0 to 99.999

7.11 TROCHOIDAL SLOT (Cycle 275, DIN/ISO: G275)

Cycle run

In conjunction with Cycle 14 **CONTOUR GEOMETRY**, this cycle facilitates the complete machining of **open** slots or contour slots using trochoidal milling.

With trochoidal milling, large cutting depths and high cutting speeds are possible because the equally distributed cutting conditions prevent wear-increasing influences on the tool. When tool inserts are used the entire cutting length is exploited to increase the attainable chip volume per tooth. Moreover, trochoidal milling is easy on the machine mechanics. Enormous amounts of time can also be saved by combining this milling method with the integrated adaptive feed control **AFC** software option (see User's Manual on conversational programming).

Depending on the cycle parameters you select, the following machining alternatives are available:

- Complete machining: Roughing, side finishing
- Only roughing
- Only side finishing

Roughing
The contour description of the open slot must always start with an approach block (**APPR**).

1. Following the positioning logic, the tool moves to the starting point of the machining operation as defined by the parameters in the **APPR** block, and positions there perpendicular to the first plunging depth.
2. The TNC roughs the slot in circular motions to the contour end point. During the circular motion the TNC moves the tool in machining direction by an infeed you can define (**Q436**). Define climb or up-cut of the circular motion in parameter **Q351**.
3. At the contour end point, the TNC moves the tool to clearance height and returns to the starting point of the contour description.
4. This process is repeated until the programmed slot depth is reached.

Finishing

5. Inasmuch as a finishing allowance is defined, the TNC finishes the slot walls, in multiple infeeds if so specified. Starting from the defined starting point of the APPR block, the TNC approaches the slot wall. Climb or up-cut are taken into consideration.

Example: TROCHOIDAL SLOT scheme

```
0 BEGIN PGM CYC275 MM
...
12 CYCL DEF 14.0 CONTOUR GEOMETRY
13 CYCL DEF 14.1 CONTOUR LABEL 10
14 CYCL DEF 275 TROCHOIDAL SLOT ...
15 CYCL CALL M3
...
50 L Z+250 R0 FMAX M2
51 LBL 10
...
55 LBL 0
...
99 END PGM CYC275 MM
```

Please note while programming:

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

When using Cycle 275 **TROCHOIDAL SLOT**, you can define only one contour program in Cycle 14 **CONTOUR GEOMETRY**.

Define the center line of the slot with all available path functions in the contour subprogram.

The memory capacity for programming an SL cycle is limited. You can program up to 4090 contour elements in one SL cycle.

The TNC does not need Cycle 20 **CONTOUR DATA** in conjunction with Cycle 275.

Machining of a closed contour is not possible with Cycle 275.

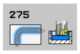
Danger of collision!

To avoid collisions,

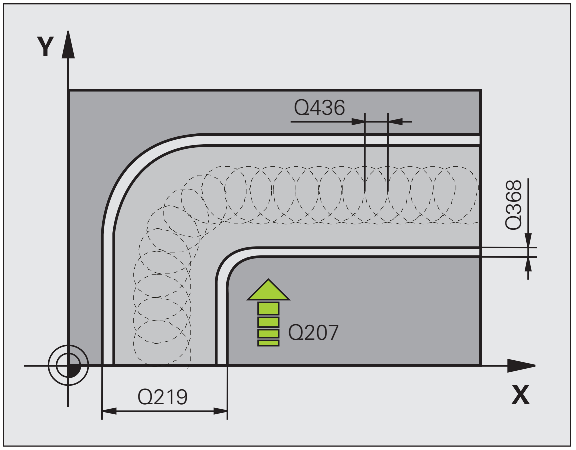
- Do not program positions in incremental dimensions immediately after Cycle 275 since they are referenced to the position of the tool at the end of the cycle.
- Move the tool to defined (absolute) positions in all main axes, since the position of the tool at the end of the cycle is not identical to the position of the tool at the start of the cycle.

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

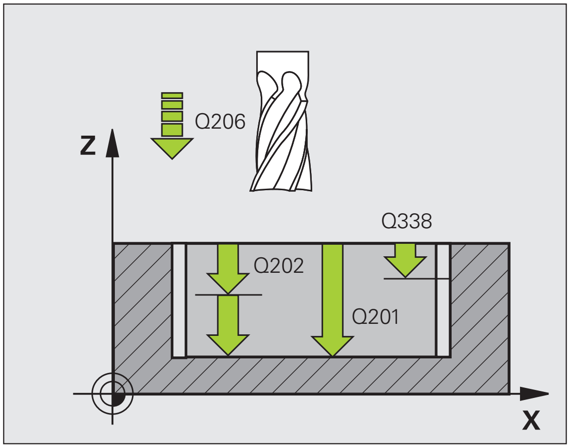
Cycle parameters



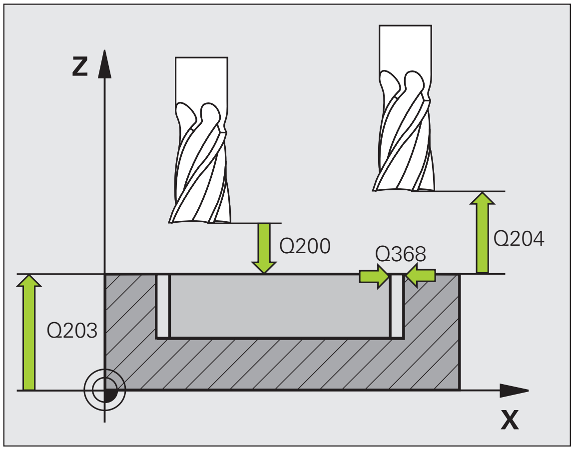
- **Machining operation (0/1/2)** Q215: Define the machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  The TNC also executes side finishing if the finishing allowance (Q368) defined is 0.
- **Slot width** Q219: Enter the slot width; if you enter a slot width that equals the tool diameter, the TNC will only machine the contour outline. Input range 0 to 99999.9999
- **Finishing allowance for side** Q368 (incremental): Finishing allowance in the working plane
- **Infeed per rev.** Q436 absolute: Value by which the TNC moves the tool in the machining direction per revolution. Input range: 0 to 99999.9999
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Climb or up-cut** Q351: Type of milling operation with M3:
  +1 = climb milling
  –1 = up-cut milling
  Alternatively **PREDEF**



- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of slot. Input range -99999.9999 to 99999.9999
- **Plunging depth** Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min while moving to depth. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Infeed for finishing** Q338 (incremental): Infeed per cut. Q338=0: Finishing in one infeed. Input range 0 to 99999.9999
- **Feed rate for finishing** Q385: Traversing speed of the tool in mm/min during side finishing. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**



- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Workpiece surface coordinate** Q203 (absolute): Absolute coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Plunging strategy** Q366: Type of plunging strategy:
  - 0 = vertical plunging. The TNC plunges perpendicularly, regardless of the plunging angle **ANGLE** defined in the tool table.
  - 1: No function
  - 2 = Reciprocating plunge. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. The TNC will otherwise display an error message.
  - Alternative: **PREDEF**



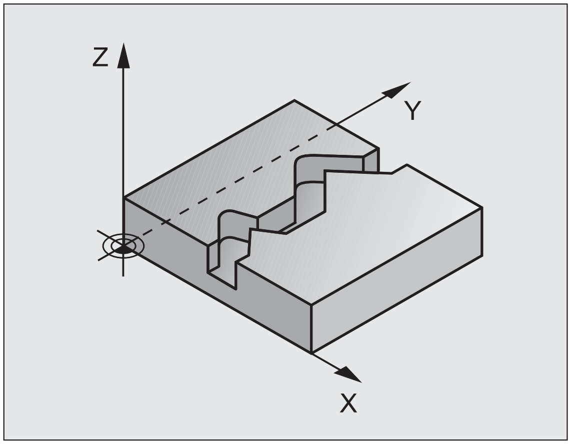
Example: NC blocks

```
8 CYCL DEF 275 TROCHOIDAL SLOT
    Q215=0     ;MACHINING OPERATION
    Q219=12    ;SLOT WIDTH
    Q368=0.2   ;ALLOWANCE FOR SIDE
    Q436=2     ;INFEED PER REVOLUTION
    Q207=500   ;FEED RATE FOR MILLING
    Q351=+1    ;CLIMB OR UP-CUT
    Q201=-20   ;DEPTH
    Q202=5     ;PLUNGING DEPTH
    Q206=150   ;FEED RATE FOR PLNGNG
    Q338=5     ;INFEED FOR FINISHING
    Q385=500   ;FEED RATE FOR FINISHING
    Q200=2     ;SET-UP CLEARANCE
    Q203=+0    ;SURFACE COORDINATE
    Q204=50    ;2ND SET-UP CLEARANCE
    Q366=2     ;PLUNGE
9 CYCL CALL FMAX M3
```

7.12 THREE-D CONTOUR TRAIN (Cycle 276, DIN/ISO: G276)

Cycle run

In conjunction with Cycle 14 **CONTOUR GEOMETRY**, this cycle facilitates the machining of open and closed contours. If necessary, you can also used the automatic identification of residual material to rework the inside corners of the contour.



Unlike Cycle 25 **CONTOUR TRAIN**, Cycle 276 **THREE-D CONTOUR TRAIN** also interprets coordinates in the tool axis (Z axis) that are defined in the contour subprogram. This makes it possible to easily machine contours created with a CAM system, for example.

Machining a contour without infeed: Milling depth Q1=0

1. Using positioning logic, the tool moves to the starting point of machining that results from the first contour point of the selected machining direction and the selected approach function.
2. The contour is approached on a tangential arc and machined up to the end.
3. When the tool reaches the end point of the contour, it departs the contour tangentially. The departure function is performed in the same manner as the approach function.
4. Finally, the TNC retracts the tool to the clearance height.

Machining a contour with infeed: Milling depth Q1 not equal to 0 and plunging depth Q10 are defined

1. Using positioning logic, the tool moves to the starting point of machining that results from the first contour point of the selected machining direction and the selected approach function.
2. The contour is approached on a tangential arc and machined up to the end.
3. When the tool reaches the end point of the contour, it departs the contour tangentially. The departure function is performed in the same manner as the approach function.
4. If reciprocating plunge is selected (**Q15=0**), the TNC moves the tool to the next plunging depth and machines the contour until the original starting point is reached. Otherwise the tool is moved to clearance height and returned to the starting point of machining. From there, the TNC moves the tool to the next plunging depth. The departure function is performed in the same manner as the approach function.
5. This process is repeated until the programmed depth is reached.
6. Finally, the TNC retracts the tool to the clearance height.

Please note while programming:

The first block in the contour subprogram must contain values in all of the three axes X, Y and Z.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the TNC will execute the cycle using the tool axis coordinates defined in the contour subprogram.

When using Cycle 25 **CONTOUR TRAIN**, you can define only one contour program in Cycle 14 **CONTOUR GEOMETRY**.

The memory capacity for programming an SL cycle is limited. You can program up to 4090 contour elements in one SL cycle.

The TNC does not need Cycle 20 **CONTOUR DATA** in conjunction with Cycle 276.

Make sure that the tool is in the tool axis above the workpiece when the cycle is called; otherwise the TNC will issue an error message.

Use the **CONTOUR TRAIN DATA** cycle to define the machining behavior of Cycle 276 (see "CONTOUR TRAIN DATA (Cycle 270, DIN/ISO: G270)" on page 204).

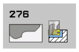
Danger of collision!

To avoid collisions,

- Before the cycle call, position the tool in the tool axis such that the TNC can approach the starting point of the contour without collision. If the actual position of the tool is below the clearance height when the cycle is called, the TNC will issue an error message.
- If you program **APPR** and **DEP** blocks for contour approach and departure, the TNC monitors whether the execution of these blocks would damage the contour.
- Do not program positions in incremental dimensions immediately after Cycle 276 since they are referenced to the position of the tool at the end of the cycle.
- Move the tool to defined (absolute) positions in all main axes, since the position of the tool at the end of the cycle is not identical to the position of the tool at the start of the cycle.

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

Cycle parameters



- **Milling depth** Q1 (incremental): Distance between workpiece surface and contour floor. If milling depth Q1 = 0 and plunging depth Q10 = 0 are programmed, the TNC machines the contour according to the Z values defined in the contour subprogram. Input range -99999.9999 to 99999.9999
- **Finishing allowance for side** Q3 (incremental): Finishing allowance in the working plane. Input range -99999.9999 to 99999.9999
- **Clearance height** Q7 (absolute): Absolute height at which the tool cannot collide with the workpiece. Position for tool retraction at the end of the cycle. Input range -99999.9999 to 99999.9999; alternatively **PREDEF**
- **Plunging depth** Q10 (incremental): Infeed per cut. Effective only when the milling depth Q1 is defined as not equal to 0. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q11: Traversing speed of the tool in the spindle axis. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for milling** Q12: Traversing speed of the tool in the working plane. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Climb/Up-cut? Up-cut = –1** Q15:
  Climb milling: Input value = +1
  Up-cut milling: Input value = –1
  To enable climb milling and up-cut milling alternately in several infeeds:Input value = 0

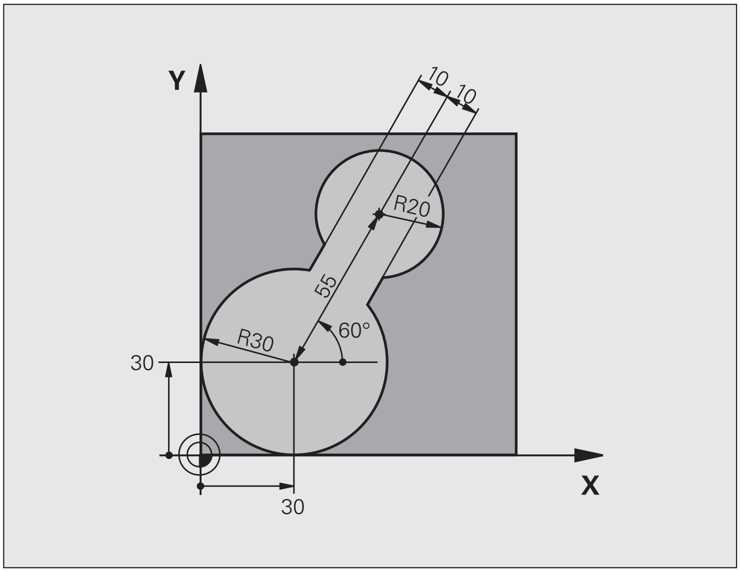
Example: NC blocks

```
62 CYCL DEF 276 THREE-D CONTOUR TRAIN
    Q1=-20    ;MILLING DEPTH
    Q3=+0     ;ALLOWANCE FOR SIDE
    Q7=+50    ;CLEARANCE HEIGHT
    Q10=+5    ;PLUNGING DEPTH
    Q11=100   ;FEED RATE FOR PLNGNG
    Q12=350   ;FEED RATE FOR MILLING
    Q15=-1    ;CLIMB OR UP-CUT
    Q18=0     ;COARSE ROUGHING TOOL
    Q446=0.01 ;RESIDUAL MATERIAL
    Q447=10   ;CONNECTION DISTANCE
    Q448=2    ;PATH EXTENSION
```

- **Coarse roughing tool** Q18 or QS18: Number or name of the tool with which the TNC has already coarse-roughed the contour. Press the TOOL NAME soft key to switch to name input. The TNC automatically inserts the closing quotation mark when you exit the input field. If there was no coarse roughing, enter "0": the TNC will machine the contour as much as is possible withe the active tool; if you enter a number or a name, the TNC will machine only the contour part that could not be machined with the coarse roughing tool. Input range 0 to 32767.9 if a number is entered; maximum 32 characters if a name is entered.
- **Accepted residual material** Q446: Remaining material thickness as of which the TNC should no longer machine the contour. Default value: 0.01 mm Input range 0 to +9.999
- **Maximum connection distance** Q447: Maximum distance between two areas to be fine-roughed, between which the tool is to move at the machining depth along the contour without a lift-off motion. Input range 0 to 999
- **Path extension** Q448: Length by which the tool path is extended at the start and end of the contour. The TNC always extends the tool path parallel to the contour. Input range 0 to 99.999

7.13 Programming examples

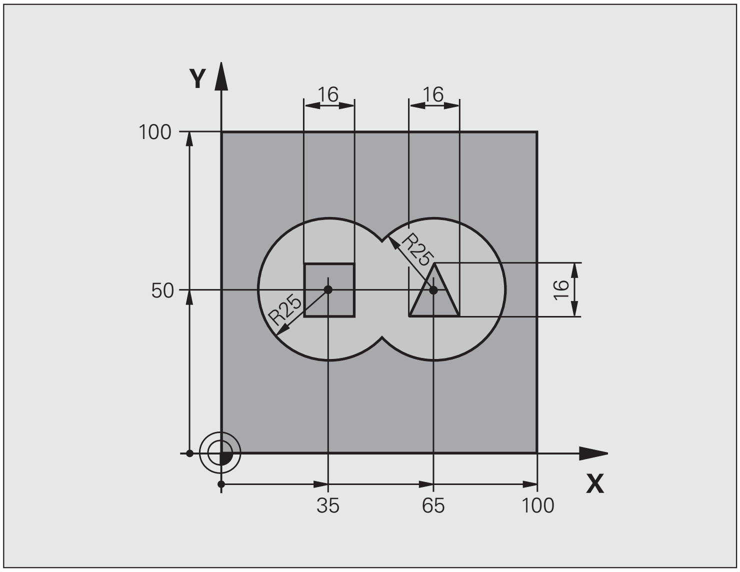
Example: Roughing-out and fine-roughing a pocket



0 BEGIN PGM C20 MM	
1 BLK FORM 0.1 Z X-10 Y-10 Z-40	
2 BLK FORM 0.2 X+100 Y+100 Z+0	Definition of workpiece blank
3 TOOL CALL 1 Z S2500	Tool call: coarse roughing tool, diameter 30
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 14.0 CONTOUR GEOMETRY	Define contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 CYCL DEF 20 CONTOUR DATA	Define general machining parameters
Q1=-20 ;MILLING DEPTH	
Q2=1 ;TOOL PATH OVERLAP	
Q3=+0 ;ALLOWANCE FOR SIDE	
Q4=+0 ;ALLOWANCE FOR FLOOR	
Q5=+0 ;SURFACE COORDINATE	
Q6=2 ;SET-UP CLEARANCE	
Q7=+100 ;CLEARANCE HEIGHT	
Q8=0.1 ;ROUNDING RADIUS	
Q9=-1 ;DIRECTION OF ROTATION	

`8 CYCL DEF 22 ROUGH-OUT`	Cycle definition: Coarse roughing
`Q10=5 ;PLUNGING DEPTH`	
`Q11=100 ;FEED RATE FOR PLNGNG`	
`Q12=350 ;FEED RATE FOR ROUGHING`	
`Q18=0 ;COARSE ROUGHING TOOL`	
`Q19=150 ;RECIPROCATION FEED RATE`	
`Q208=30000;RETRACTION FEED RATE`	
`Q401=100 ;FEED RATE FACTOR`	
`Q404=0 ;FINE ROUGH STRATEGY`	
`9 CYCL CALL M3`	Cycle call: Coarse roughing
`10 L Z+250 R0 FMAX M6`	Tool change
`11 TOOL CALL 2 Z S3000`	Tool call: fine roughing tool, diameter 15
`12 CYCL DEF 22 ROUGH-OUT`	Define the fine roughing cycle
`Q10=5 ;PLUNGING DEPTH`	
`Q11=100 ;FEED RATE FOR PLNGNG`	
`Q12=350 ;FEED RATE FOR ROUGHING`	
`Q18=1 ;COARSE ROUGHING TOOL`	
`Q19=150 ;RECIPROCATION FEED RATE`	
`Q208=30000;RETRACTION FEED RATE`	
`Q401=100 ;FEED RATE FACTOR`	
`Q404=0 ;FINE ROUGH STRATEGY`	
`13 CYCL CALL M3`	Cycle call: Fine roughing
`14 L Z+250 R0 FMAX M2`	Retract the tool, end program
`15 LBL 1`	Contour subprogram
`16 L X+0 Y+30 RR`	
`17 FC DR- R30 CCX+30 CCY+30`	
`18 FL AN+60 PDX+30 PDY+30 D10`	
`19 FSELECT 3`	
`20 FPOL X+30 Y+30`	
`21 FC DR- R20 CCPR+55 CCPA+60`	
`22 FSELECT 2`	
`23 FL AN-120 PDX+30 PDY+30 D10`	
`24 FSELECT 3`	
`25 FC X+0 DR- R30 CCX+30 CCY+30`	
`26 FSELECT 2`	
`27 LBL 0`	
`28 END PGM C20 MM`	

Example: Pilot drilling, roughing-out and finishing overlapping contours

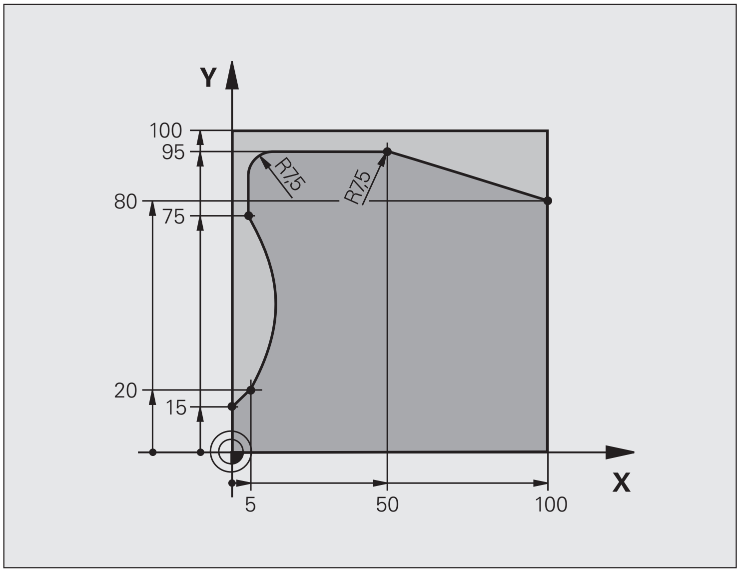


0 BEGIN PGM C21 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S2500	Tool call: Drill, diameter 12
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 14.0 CONTOUR GEOMETRY	Define contour subprograms
6 CYCL DEF 14.1 CONTOUR LABEL 1/2/3/4	
7 CYCL DEF 20 CONTOUR DATA	Define general machining parameters
Q1=-20 ;MILLING DEPTH	
Q2=1 ;TOOL PATH OVERLAP	
Q3=+0.5 ;ALLOWANCE FOR SIDE	
Q4=+0.5 ;ALLOWANCE FOR FLOOR	
Q5=+0 ;SURFACE COORDINATE	
Q6=2 ;SET-UP CLEARANCE	
Q7=+100 ;CLEARANCE HEIGHT	
Q8=0.1 ;ROUNDING RADIUS	
Q9=-1 ;DIRECTION OF ROTATION	

8 CYCL DEF 21 PILOT DRILLING	Cycle definition: Pilot drilling
Q10=5 ;PLUNGING DEPTH	
Q11=250 ;FEED RATE FOR PLNGNG	
Q13=2 ;ROUGH-OUT TOOL	
9 CYCL CALL M3	Cycle call: Pilot drilling
10 L +250 R0 FMAX M6	Tool change
11 TOOL CALL 2 Z S3000	Call the tool for roughing/finishing, diameter 12
12 CYCL DEF 22 ROUGH-OUT	Cycle definition: Rough-out
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=350 ;FEED RATE FOR ROUGHING	
Q18=0 ;COARSE ROUGHING TOOL	
Q19=150 ;RECIPROCATION FEED RATE	
Q208=30000;RETRACTION FEED RATE	
Q401=100 ;FEED RATE FACTOR	
Q404=0 ;FINE ROUGH STRATEGY	
13 CYCL CALL M3	Cycle call: Rough-out
14 CYCL DEF 23 FLOOR FINISHING	Cycle definition: Floor finishing
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=200 ;FEED RATE FOR ROUGHING	
Q208=30000;RETRACTION FEED RATE	
15 CYCL CALL	Cycle call: Floor finishing
16 CYCL DEF 24 SIDE FINISHING	Cycle definition: Side finishing
Q9=+1 ;DIRECTION OF ROTATION	
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=400 ;FEED RATE FOR ROUGHING	
Q14=+0 ;ALLOWANCE FOR SIDE	
17 CYCL CALL	Cycle call: Side finishing
18 L Z+250 R0 FMAX M2	Retract the tool, end program

19 LBL 1	Contour subprogram 1: left pocket
20 CC X+35 Y+50	
21 L X+10 Y+50 RR	
22 C X+10 DR-	
23 LBL 0	
24 LBL 2	Contour subprogram 2: right pocket
25 CC X+65 Y+50	
26 L X+90 Y+50 RR	
27 C X+90 DR-	
28 LBL 0	
29 LBL 3	Contour subprogram 3: square left island
30 L X+27 Y+50 RL	
31 L Y+58	
32 L X+43	
33 L Y+42	
34 L X+27	
35 LBL 0	
36 LBL 4	Contour subprogram 4: triangular right island
39 L X+65 Y+42 RL	
37 L X+57	
38 L X+65 Y+58	
39 L X+73 Y+42	
40 LBL 0	
41 END PGM C21 MM	

Example: Contour train



`0 BEGIN PGM C25 MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-40`	Definition of workpiece blank
`2 BLK FORM 0.2 X+100 Y+100 Z+0`	
`3 TOOL CALL 1 Z S2000`	Tool call: Diameter 20
`4 L Z+250 R0 FMAX`	Retract the tool
`5 CYCL DEF 14.0 CONTOUR GEOMETRY`	Define contour subprogram
`6 CYCL DEF 14.1 CONTOUR LABEL 1`	
`7 CYCL DEF 25 CONTOUR TRAIN`	Define machining parameters
`Q1=-20 ;MILLING DEPTH`	
`Q3=+0 ;ALLOWANCE FOR SIDE`	
`Q5=+0 ;SURFACE COORDINATE`	
`Q7=+250 ;CLEARANCE HEIGHT`	
`Q10=5 ;PLUNGING DEPTH`	
`Q11=100 ;FEED RATE FOR PLNGNG`	
`Q12=200 ;FEED RATE FOR MILLING`	
`Q15=+1 ;CLIMB OR UP-CUT`	
`8 CYCL CALL M3`	Cycle call
`9 L Z+250 R0 FMAX M2`	Retract the tool, end program

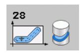
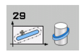
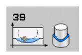
10 LBL 1	Contour subprogram
11 L X+0 Y+15 RL	
12 L X+5 Y+20	
13 CT X+5 Y+75	
14 L Y+95	
15 RND R7.5	
16 L X+50	
17 RND R7.5	
18 L X+100 Y+80	
19 LBL 0	
20 END PGM C25 MM	

8

Fixed Cycles: Cylindrical Surface

8.1 Fundamentals

Overview of cylindrical surface cycles

Cycle	Soft key	Page
27 CYLINDER SURFACE		Page 229
28 CYLINDER SURFACE slot milling		Page 232
29 CYLINDER SURFACE ridge milling		Page 235
39 CYLINDER SURFACE outside contour milling		Page 238

8.2 CYLINDER SURFACE (Cycle 27, DIN/ISO: G127, software option 1)

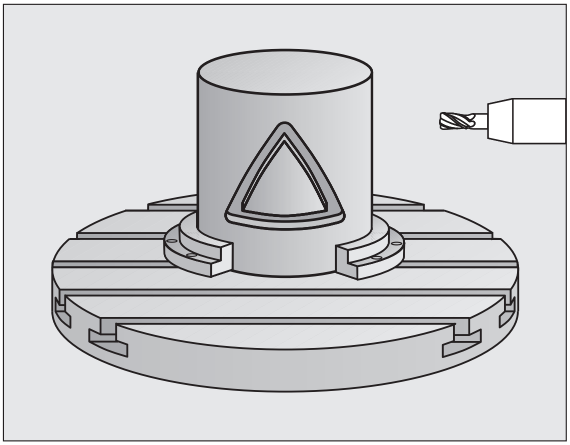
Cycle run

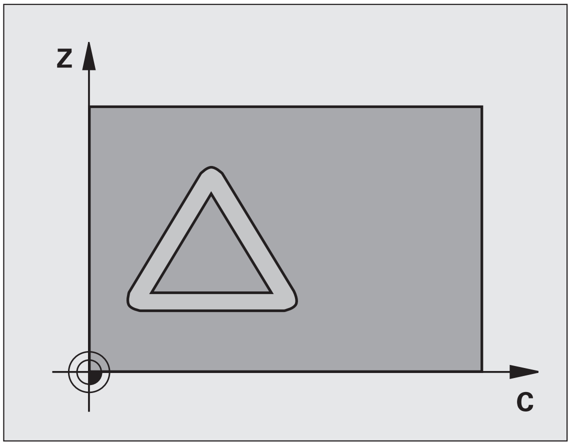
This cycle enables you to program a contour in two dimensions and then roll it onto a cylindrical surface for 3-D machining. Use Cycle 28 if you want to mill guideways on the cylinder.

The contour is described in a subprogram identified in Cycle 14 CONTOUR.

The subprogram contains coordinates in a rotary axis and in its parallel axis. The rotary axis C, for example, is parallel to the Z axis. The path functions **L**, **CHF**, **CR**, **RND**, **APPR** (except **APPR LCT**) and **DEP** are available.

The dimensions in the rotary axis can be entered as desired either in degrees or in mm (or inches). You can select the desired dimension type in the cycle definition.





1. The TNC positions the tool over the cutter infeed point, taking the allowance for side into account.
2. At the first plunging depth, the tool mills along the programmed contour at the milling feed rate Q12.
3. At the end of the contour, the TNC returns the tool to the set-up clearance and returns to the point of penetration.
4. Steps 1 to 3 are repeated until the programmed milling depth Q1 is reached.
5. Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle (depending on MP7420).

Please note while programming:

The machine and TNC must be prepared for cylinder surface interpolation by the machine tool builder. Refer to your machine manual.

In the first NC block of the contour subprogram, always program both cylinder surface coordinates.

The memory capacity for programming an SL cycle is limited. You can program up to 8192 contour elements in one SL cycle.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

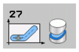
This cycle requires a center-cut end mill (ISO 1641).

The cylinder must be set up centered on the rotary table.

The tool axis must be perpendicular to the rotary table. If this is not the case, the TNC will generate an error message.

This cycle can also be used in a tilted working plane.

Cycle parameters



- **Milling depth** Q1 (incremental): Distance between the cylindrical surface and the floor of the contour. Input range -99999.9999 to 99999.9999
- **Finishing allowance for side** Q3 (incremental): Finishing allowance in the plane of the unrolled cylindrical surface. This allowance is effective in the direction of the radius compensation. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q6 (incremental): Distance between the tool tip and the cylinder surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Plunging depth** Q10 (incremental): Infeed per cut. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q11: Traversing speed of the tool in the spindle axis. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for milling** Q12: Traversing speed of the tool in the working plane. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Cylinder radius** Q16: Radius of the cylinder on which the contour is to be machined. Input range 0 to 99999.9999
- **Dimension type? Degrees=0 MM/INCH=1** Q17: The dimensions for the rotary axis of the subprogram are given either in degrees (0) or in mm/inches (1)

Example: NC blocks

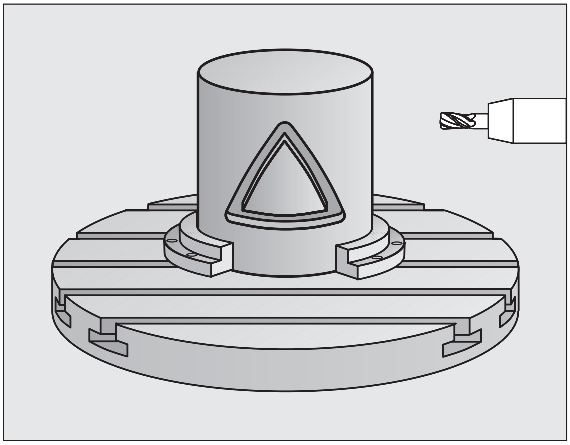
```
63 CYCL DEF 27 CYLINDER SURFACE
    Q1=-8      ;MILLING DEPTH
    Q3=+0      ;ALLOWANCE FOR SIDE
    Q6=+0      ;SET-UP CLEARANCE
    Q10=+3     ;PLUNGING DEPTH
    Q11=100    ;FEED RATE FOR PLNGNG
    Q12=350    ;FEED RATE FOR MILLING
    Q16=25     ;RADIUS
    Q17=0      ;TYPE OF DIMENSION
```

8.3 CYLINDER SURFACE slot milling (Cycle 28, DIN/ISO: G128, software option 1)

Cycle run

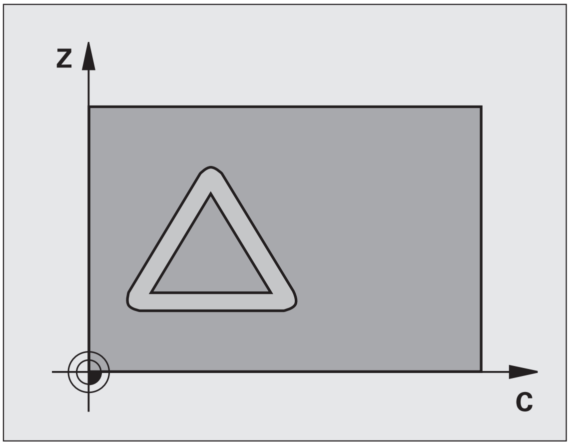
This cycle enables you to program a guide notch in two dimensions and then transfer it onto a cylindrical surface. Unlike Cycle 27, with this cycle the TNC adjusts the tool so that, with radius compensation active, the walls of the slot are nearly parallel. You can machine exactly parallel walls by using a tool that is exactly as wide as the slot.

The smaller the tool is with respect to the slot width, the larger the distortion in circular arcs and oblique line segments. To minimize this process-related distortion, you can define in parameter Q21 a tolerance with which the TNC machines a slot as similar as possible to a slot machined with a tool of the same width as the slot.



Program the midpoint path of the contour together with the tool radius compensation. With the radius compensation you specify whether the TNC cuts the slot with climb milling or up-cut milling.

1 The TNC positions the tool over the cutter infeed point.
2 At the first plunging depth, the tool mills along the programmed slot wall at the milling feed rate Q12 while respecting the finishing allowance for the side.
3 At the end of the contour, the TNC moves the tool to the opposite wall and returns to the infeed point.
4 Steps 2 and 3 are repeated until the programmed milling depth Q1 is reached.
5 If you have defined the tolerance in Q21, the TNC then remachines the slot walls to be as parallel as possible.
6 Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle (depending on MP7420).



Please note while programming:

The machine and TNC must be prepared for cylinder surface interpolation by the machine tool builder. Refer to your machine manual.

In the first NC block of the contour subprogram, always program both cylinder surface coordinates.

The memory capacity for programming an SL cycle is limited. You can program up to 8192 contour elements in one SL cycle.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

This cycle requires a center-cut end mill (ISO 1641).

The cylinder must be set up centered on the rotary table.

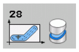
The tool axis must be perpendicular to the rotary table. If this is not the case, the TNC will generate an error message.

This cycle can also be used in a tilted working plane.

Danger of collision!

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

Cycle parameters



- **Milling depth** Q1 (incremental): Distance between the cylindrical surface and the floor of the contour. Input range -99999.9999 to 99999.9999
- **Finishing allowance for side** Q3 (incremental): Finishing allowance on the slot wall. The finishing allowance reduces the slot width by twice the entered value. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q6 (incremental): Distance between the tool tip and the cylinder surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Plunging depth** Q10 (incremental): Infeed per cut. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q11: Traversing speed of the tool in the spindle axis. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for milling** Q12: Traversing speed of the tool in the working plane. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Cylinder radius** Q16: Radius of the cylinder on which the contour is to be machined. Input range 0 to 99999.9999
- **Dimension type? Degrees=0 MM/INCH=1** Q17: The dimensions for the rotary axis of the subprogram are given either in degrees (0) or in mm/inches (1)
- **Slot width** Q20: Width of the slot to be machined. Input range -99999.9999 to 99999.9999
- **Tolerance?** Q21: If you use a tool smaller than the programmed slot width Q20, process-related distortion occurs on the slot wall wherever the slot follows the path of an arc or oblique line. If you define the tolerance Q21, the TNC adds a subsequent milling operation to ensure that the slot dimensions are as close as possible to those of a slot that has been milled with a tool exactly as wide as the slot. With Q21 you define the permitted deviation from this ideal slot. The number of subsequent milling operations depends on the cylinder radius, the tool used, and the slot depth. The smaller the tolerance is defined, the more exact the slot is and the longer the remachining takes. **Recommendation**: Use a tolerance of 0.02 mm. **Function inactive**: Enter 0 (default setting) Input range 0 to 9.9999

Example: NC blocks

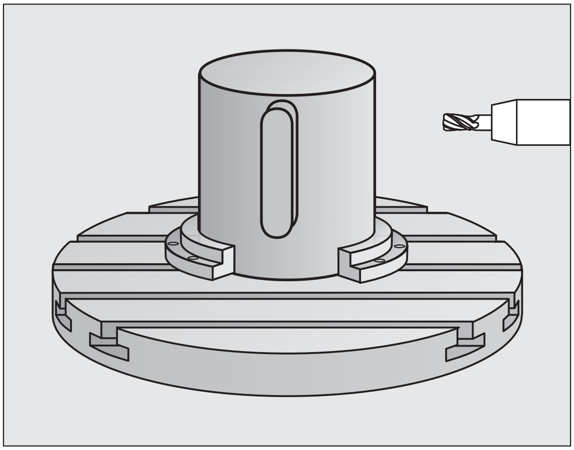
```
63 CYCL DEF 28 CYLINDER SURFACE
    Q1=-8      ;MILLING DEPTH
    Q3=+0      ;ALLOWANCE FOR SIDE
    Q6=+0      ;SET-UP CLEARANCE
    Q10=+3     ;PLUNGING DEPTH
    Q11=100    ;FEED RATE FOR PLNGNG
    Q12=350    ;FEED RATE FOR MILLING
    Q16=25     ;RADIUS
    Q17=0      ;TYPE OF DIMENSION
    Q20=12     ;SLOT WIDTH
    Q21=0      ;TOLERANCE
```

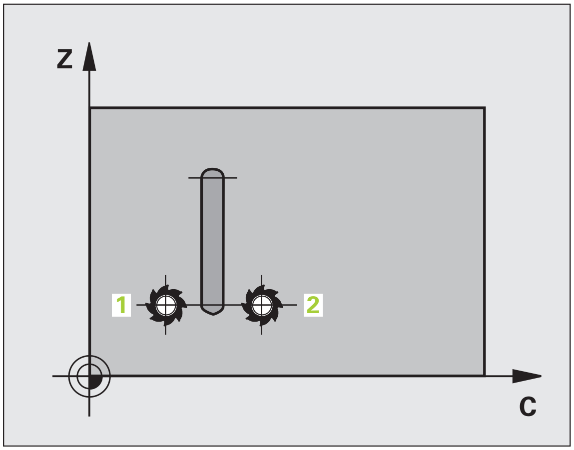
8.4 CYLINDER SURFACE ridge milling (Cycle 29, DIN/ISO: G129, software option 1)

Cycle run

This cycle enables you to program a ridge in two dimensions and then transfer it onto a cylindrical surface. With this cycle the TNC adjusts the tool so that, with radius compensation active, the walls of the slot are always parallel. Program the midpoint path of the ridge together with the tool radius compensation. With the radius compensation you specify whether the TNC cuts the ridge with climb milling or up-cut milling.

At the ends of the ridge the TNC always adds a semicircle whose radius is half the ridge width.





1. The TNC positions the tool over the starting point of machining. The TNC calculates the starting point from the ridge width and the tool diameter. It is located next to the first point defined in the contour subprogram, offset by half the ridge width and the tool diameter. The radius compensation determines whether machining begins from the left (**1**, RL = climb milling) or the right of the ridge (**2**, RR = up-cut milling).
2. After the TNC has positioned to the first plunging depth, the tool moves on a circular arc at the milling feed rate Q12 tangentially to the ridge wall. A finishing allowance programmed for the side is taken into account.
3. At the first plunging depth, the tool mills along the programmed ridge wall at the milling feed rate Q12 until the stud is completed.
4. The tool then departs the ridge wall on a tangential path and returns to the starting point of machining.
5. Steps 2 to 4 are repeated until the programmed milling depth Q1 is reached.
6. Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle (depending on MP7420).

Please note while programming:

The machine and TNC must be prepared for cylinder surface interpolation by the machine tool builder. Refer to your machine manual.

In the first NC block of the contour subprogram, always program both cylinder surface coordinates.

Ensure that the tool has enough space laterally for contour approach and departure.

The memory capacity for programming an SL cycle is limited. You can program up to 8192 contour elements in one SL cycle.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

The cylinder must be set up centered on the rotary table.

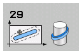
The tool axis must be perpendicular to the rotary table. If this is not the case, the TNC will generate an error message.

This cycle can also be used in a tilted working plane.

Danger of collision!

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

Cycle parameters



- **Milling depth** Q1 (incremental): Distance between the cylindrical surface and the floor of the contour. Input range -99999.9999 to 99999.9999
- **Finishing allowance for side** Q3 (incremental): Finishing allowance on the ridge wall. The finishing allowance increases the ridge width by twice the entered value. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q6 (incremental): Distance between the tool tip and the cylinder surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Plunging depth** Q10 (incremental): Infeed per cut. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q11: Traversing speed of the tool in the spindle axis. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for milling** Q12: Traversing speed of the tool in the working plane. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Cylinder radius** Q16: Radius of the cylinder on which the contour is to be machined. Input range 0 to 99999.9999
- **Dimension type? Degrees=0 MM/INCH=1** Q17: The dimensions for the rotary axis of the subprogram are given either in degrees (0) or in mm/inches (1)
- **Ridge width** Q20: Width of the ridge to be machined. Input range -99999.9999 to 99999.9999

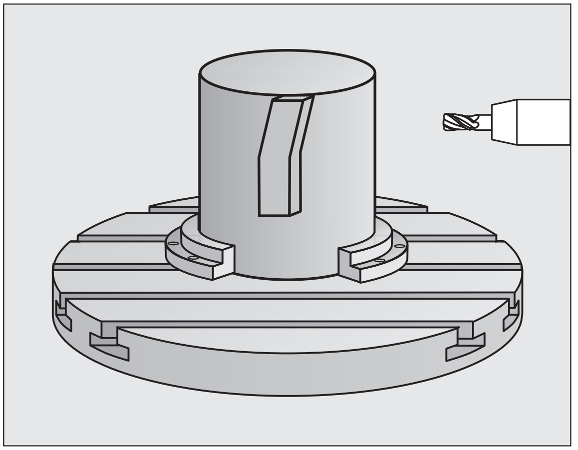
Example: NC blocks

```
63 CYCL DEF 29 CYLINDER SURFACE RIDGE
    Q1=-8      ;MILLING DEPTH
    Q3=+0      ;ALLOWANCE FOR SIDE
    Q6=+0      ;SET-UP CLEARANCE
    Q10=+3     ;PLUNGING DEPTH
    Q11=100    ;FEED RATE FOR PLNGNG
    Q12=350    ;FEED RATE FOR MILLING
    Q16=25     ;RADIUS
    Q17=0      ;TYPE OF DIMENSION
    Q20=12     ;RIDGE WIDTH
```

8.5 CYLINDER SURFACE outside contour milling (Cycle 39, DIN/ISO: G139, software option 1)

Cycle run

This cycle enables you to program an open contour in two dimensions and then roll it onto a cylindrical surface for 3-D machining. With this cycle the TNC adjusts the tool so that, with radius compensation active, the wall of the open contour is always parallel to the cylinder axis.



Unlike Cycles 28 and 29, in the contour subprogram you define the actual contour to be machined.

1. The TNC positions the tool over the starting point of machining. The TNC locates the starting point next to the first point defined in the contour subprogram, offset by the tool diameter (standard behavior)
2. After the TNC has positioned to the first plunging depth, the tool moves on a circular arc at the milling feed rate Q12 tangentially to the contour. A finishing allowance programmed for the side is taken into account.
3. At the first plunging depth, the tool mills along the programmed contour at the milling feed rate Q12 until the contour train is completed.
4. The tool then departs the ridge wall on a tangential path and returns to the starting point of machining.
5. Steps 2 to 4 are repeated until the programmed milling depth Q1 is reached.
6. Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle (depending on MP7420).

You can define the approach behavior of Cycle 39 in MP7680, bit 16.

- Bit 16 = 0:
  Tangential approach and departure
- Bit 16 = 1:
  Move to depth vertically at the starting point of the contour without tangential tool approach and move up at the contour end point without tangential departure.

Please note while programming:

The machine and TNC must be prepared for cylinder surface interpolation by the machine tool builder. Refer to your machine manual.

In the first NC block of the contour subprogram, always program both cylinder surface coordinates.

Ensure that the tool has enough space laterally for contour approach and departure.

The memory capacity for programming an SL cycle is limited. You can program up to 8192 contour elements in one SL cycle.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

The cylinder must be set up centered on the rotary table.

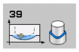
The tool axis must be perpendicular to the rotary table. If this is not the case, the TNC will generate an error message.

This cycle can also be used in a tilted working plane.

Danger of collision!

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if the spindle is not running when the cycle is called. The function also needs to be adapted by your machine manufacturer.

Cycle parameters



- **Milling depth** Q1 (incremental): Distance between the cylindrical surface and the floor of the contour. Input range -99999.9999 to 99999.9999
- **Finishing allowance for side** Q3 (incremental): Finishing allowance on the contour wall. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q6 (incremental): Distance between the tool tip and the cylinder surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Plunging depth** Q10 (incremental): Infeed per cut. Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** Q11: Traversing speed of the tool in the spindle axis. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for milling** Q12: Traversing speed of the tool in the working plane. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Cylinder radius** Q16: Radius of the cylinder on which the contour is to be machined. Input range 0 to 99999.9999
- **Dimension type? Degrees=0 MM/INCH=1** Q17: The dimensions for the rotary axis of the subprogram are given either in degrees (0) or in mm/inches (1)

Example: NC blocks

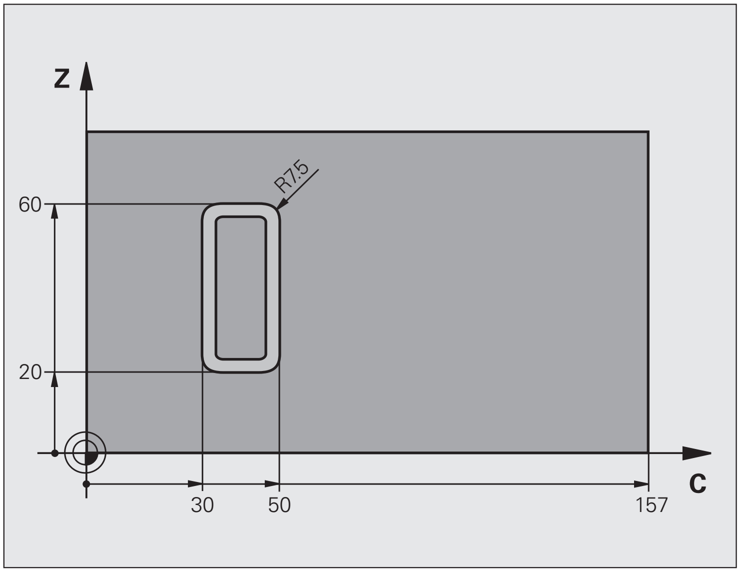
```
63 CYCL DEF 39 CYL. SURFACE CONTOUR
    Q1=-8       ;MILLING DEPTH
    Q3=+0       ;ALLOWANCE FOR SIDE
    Q6=+0       ;SET-UP CLEARANCE
    Q10=+3      ;PLUNGING DEPTH
    Q11=100     ;FEED RATE FOR PLNGNG
    Q12=350     ;FEED RATE FOR MILLING
    Q16=25      ;RADIUS
    Q17=0       ;TYPE OF DIMENSION
```

8.6 Programming examples

Example: Cylinder surface with Cycle 27

Note:

- Machine with B head and C table
- Cylinder centered on rotary table
- Datum at center of rotary table



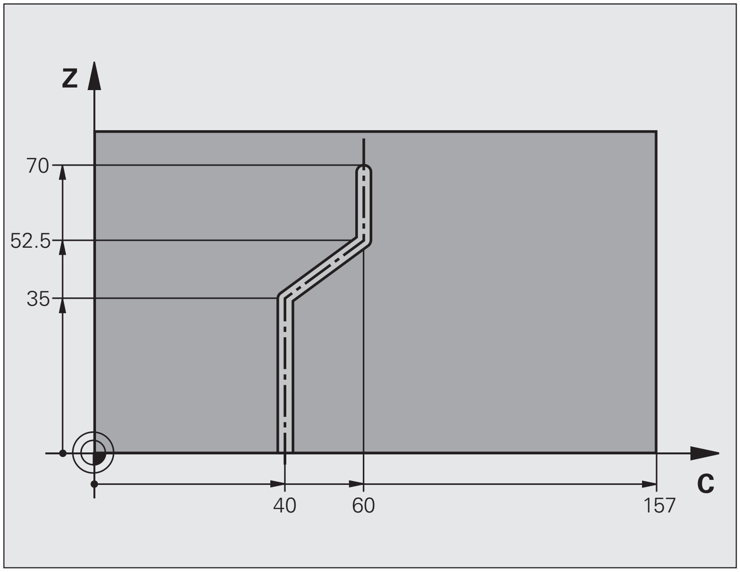
0 BEGIN PGM C27 MM	
1 TOOL CALL 1 Z S2000	Tool call: Diameter 7
2 L Z+250 R0 FMAX	Retract the tool
3 L X+50 Y0 R0 FMAX	Pre-position tool at rotary table center
4 PLANE SPATIAL SPA+0 SPB+90 SPC+0 TURN MBMAX FMAX	Tilting
5 CYCL DEF 14.0 CONTOUR GEOMETRY	Define contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 CYCL DEF 27 CYLINDER SURFACE	Define machining parameters
Q1=-7 ;MILLING DEPTH	
Q3=+0 ;ALLOWANCE FOR SIDE	
Q6=2 ;SET-UP CLEARANCE	
Q10=4 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=250 ;FEED RATE FOR MILLING	
Q16=25 ;RADIUS	
Q17=1 ;TYPE OF DIMENSION	

`8 L C+0 R0 FMAX M13 M99`	Pre-position rotary table, spindle ON, call the cycle
`9 L Z+250 R0 FMAX`	Retract the tool
`10 PLANE RESET TURN FMAX`	Tilt back, cancel the PLANE function
`11 M2`	End of program
`12 LBL 1`	Contour subprogram
`13 L C+40 X+20 RL`	Data for the rotary axis are entered in mm (Q17=1), traverse in the X axis because of 90° tilting
`14 L C+50`	
`15 RND R7.5`	
`16 L X+60`	
`17 RND R7.5`	
`18 L IC-20`	
`19 RND R7.5`	
`20 L X+20`	
`21 RND R7.5`	
`22 L C+40`	
`23 LBL 0`	
`24 END PGM C27 MM`	

Example: Cylinder surface with Cycle 28

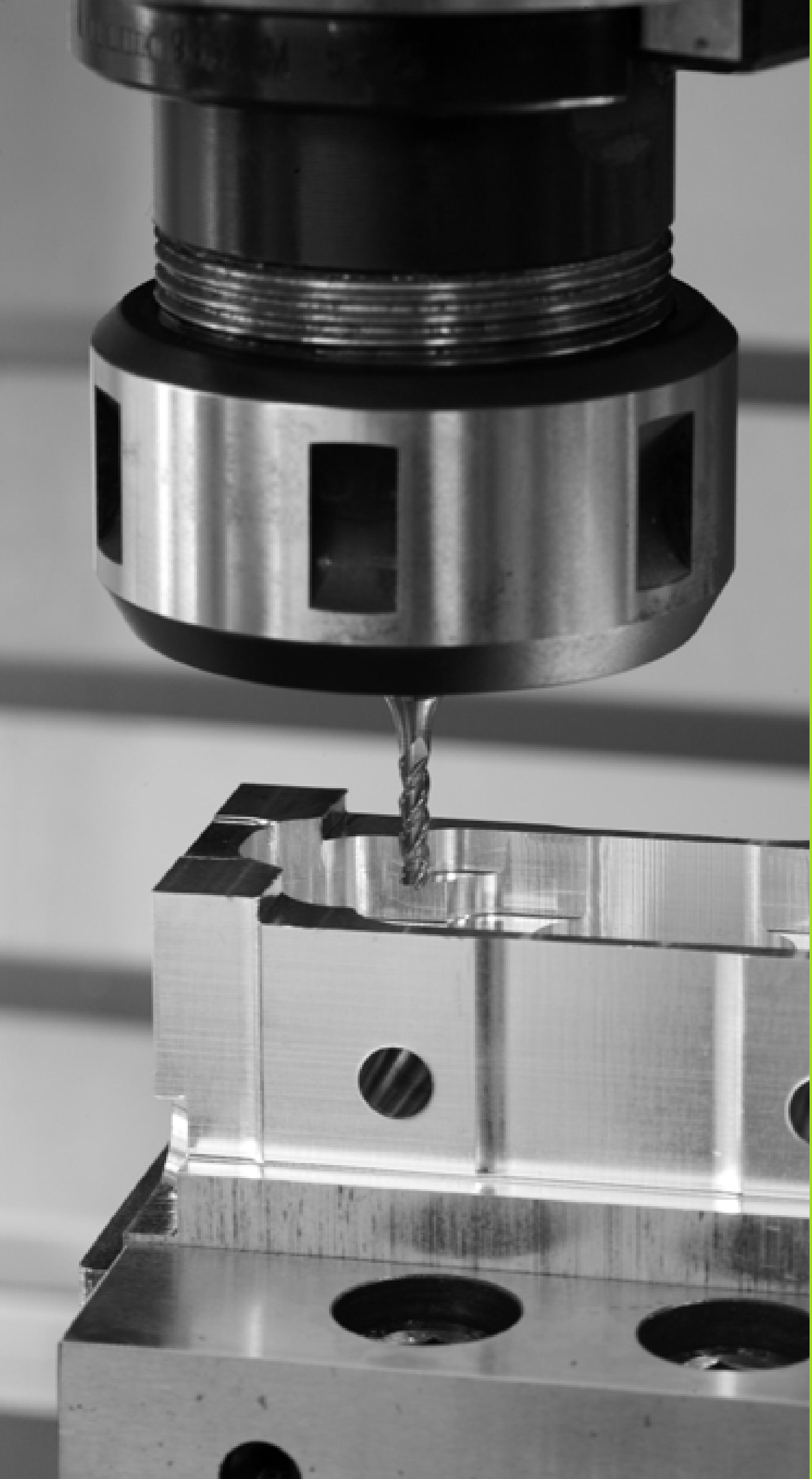
Notes:

- Cylinder centered on rotary table
- Machine with B head and C table
- Datum at center of rotary table
- Description of the midpoint path in the contour subprogram



0 BEGIN PGM C28 MM	
1 TOOL CALL 1 Z S2000	Tool call, tool axis Z, diameter 7
2 L Z+250 R0 FMAX	Retract the tool
3 L X+50 Y+0 R0 FMAX	Position tool at rotary table center
4 PLANE SPATIAL SPA+0 SPB+90 SPC+0 TURN FMAX	Tilting
5 CYCL DEF 14.0 CONTOUR GEOMETRY	Define contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 CYCL DEF 28 CYLINDER SURFACE	Define machining parameters
Q1=-7 ;MILLING DEPTH	
Q3=+0 ;ALLOWANCE FOR SIDE	
Q6=2 ;SET-UP CLEARANCE	
Q10=-4 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=250 ;FEED RATE FOR MILLING	
Q16=25 ;RADIUS	
Q17=1 ;TYPE OF DIMENSION	
Q20=10 ;SLOT WIDTH	
Q21=0.02 ;TOLERANCE	Remachining active

`8 L C+0 R0 FMAX M3 M99`	Pre-position rotary table, spindle ON, call the cycle
`9 L Z+250 R0 FMAX`	Retract the tool
`10 PLANE RESET TURN FMAX`	Tilt back, cancel the PLANE function
`11 M2`	End of program
`12 LBL 1`	Contour subprogram, description of the midpoint path
`13 L C+40 X+0 RL`	Data for the rotary axis are entered in mm (Q17=1), traverse in the X axis because of 90° tilting
`14 L X+35`	
`15 L C+60 X+52.5`	
`16 L X+70`	
`17 LBL 0`	
`18 END PGM C28 MM`	



9

Fixed Cycles: Contour Pocket with Contour Formula

9.1 SL cycles with complex contour formula

Fundamentals

SL cycles and the complex contour formula enable you to form complex contours by combining subcontours (pockets or islands). You define the individual subcontours (geometry data) as separate programs. In this way, any subcontour can be used any number of times. The TNC calculates the complete contour from the selected subcontours, which you link together through a contour formula.

The memory capacity for programming an SL cycle (all contour description programs) is limited to **128 contours.** The number of possible contour elements depends on the type of contour (inside or outside contour) and the number of contour descriptions. You can program up to **8192** elements.

The SL cycles with contour formula presuppose a structured program layout and enable you to save frequently used contours in individual programs. Using the contour formula, you can connect the subcontours to a complete contour and define whether it applies to a pocket or island.

In its present form, the "SL cycles with contour formula" function requires input from several areas in the TNC's user interface. This function is to serve as a basis for further development.

Example: Program structure: Machining with SL cycles and complex contour formula

```
0 BEGIN PGM CONTOUR MM
...
5 SEL CONTOUR "MODEL"
6 CYCL DEF 20 CONTOUR DATA ...
8 CYCL DEF 22 ROUGH-OUT ...
9 CYCL CALL
...
12 CYCL DEF 23 FLOOR FINISHING ...
13 CYCL CALL
...
16 CYCL DEF 24 SIDE FINISHING ...
17 CYCL CALL
63 L Z+250 R0 FMAX M2
64 END PGM CONTOUR  MM
```

Properties of the subcontours

- By default, the TNC assumes that the contour is a pocket. Do not program a radius compensation. In the contour formula you can convert a pocket to an island by making it negative.
- The TNC ignores feed rates F and miscellaneous functions M.
- Coordinate transformations are allowed. If they are programmed within the subcontour they are also effective in the following subprograms, but they need not be reset after the cycle call.
- Although the subprograms can contain coordinates in the spindle axis, such coordinates are ignored.
- The working plane is defined in the first coordinate block of the subprogram. The secondary axes U,V,W are permitted.

Characteristics of the fixed cycles

- The TNC automatically positions the tool to the set-up clearance before a cycle.
- Each level of infeed depth is milled without interruptions since the cutter traverses around islands instead of over them.
- The radius of "inside corners" can be programmed—the tool keeps moving to prevent surface blemishes at inside corners (this applies to the outermost pass in the Rough-out and Side Finishing cycles).
- The contour is approached in a tangential arc for side finishing.
- For floor finishing, the tool again approaches the workpiece on a tangential arc (for tool axis Z, for example, the arc is in the Z/X plane).
- The contour is machined throughout in either climb or up-cut milling.

With Machine Parameter 7420 you can determine where the tool is positioned at the end of Cycles 21 to 24.

The machining data (such as milling depth, finishing allowance and set-up clearance) are entered as CONTOUR DATA in Cycle 20.

Example: Program structure: Calculation of the subcontours with contour formula

```
0 BEGIN PGM MODEL MM
1 DECLARE CONTOUR QC1 = "CIRCLE1"
2 DECLARE CONTOUR QC2 = "CIRCLE31XY"
3 DECLARE CONTOUR QC3 = "TRIANGLE"
4 DECLARE CONTOUR QC4 = "SQUARE"
5 QC10 = ( QC1 | QC3 | QC4 ) \ QC2
6 END PGM MODEL MM

0 BEGIN PGM CIRCLE1 MM
1 CC X+75 Y+50
2 LP PR+45 PA+0
3 CP IPA+360 DR+
4 END PGM CIRCLE1 MM

0 BEGIN PGM CIRCLE31XY MM
...
...
```

Selecting a program with contour definitions

With the **SEL CONTOUR** function you select a program with contour definitions, from which the TNC takes the contour descriptions:

SPEC FCT
- Show the soft-key row with special functions

CONTOUR + POINT MACHINING
- Select the menu for functions for contour and point machining

COMPLEX CONTOUR FORMULAS
- Select the COMPLEX CONTOUR FORMULA menu

SEL CONTOUR
- Press the SEL CONTOUR soft key

SELECTION WINDOW
- Press the WINDOW SELECTION soft key: The TNC superimposes a window where you can select the program with the contour definitions
- Select a program with the arrow keys or by mouse click and confirm by pressing ENT: The TNC enters the complete path name in the **SEL CONTOUR** block
- Conclude this function with the END key
- Enter the full name of the program with the contour definition and confirm with the END key

Alternatively you can also enter the program name or the complete path name of the program with the contour definition directly via the keyboard.

Program a **SEL CONTOUR** block before the SL cycles. Cycle **14 CONTOUR GEOMETRY** is no longer necessary if you use **SEL CONTUR**.

Defining contour descriptions

With the **DECLARE CONTOUR** function you enter in a program the path for programs from which the TNC draws the contour descriptions. In addition, you can select a separate depth for this contour description (FCL 2 function):

SPEC FCT
- Show the soft-key row with special functions

CONTOUR + POINT MACHINING
- Select the menu for functions for contour and point machining

COMPLEX CONTOUR FORMULAS
- Select the COMPLEX CONTOUR FORMULA menu

DECLARE CONTOUR
- Press the DECLARE CONTOUR soft key
- Enter the number for the contour designator **QC**, and confirm with the ENT key

SELECTION WINDOW
- Press the WINDOW SELECTION soft key: The TNC superimposes a window where you can select the program to be called
- Select the program with the contour description with the arrow keys or by mouse click, and confirm by pressing ENT: The TNC enters the complete path name in the **DECLARE CONTOUR** block
- Define a separate depth for the selected contour
- Conclude this function with the END key

Alternatively you can also enter the name of the program with the contour description or the complete path name of the program directly via the keyboard.

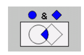
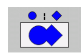
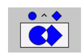
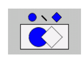
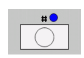
With the entered contour designators **QC** you can include the various contours in the contour formula.

If you program separate depths for contours, then you must assign a depth to all subcontours (assign the depth 0 if necessary).

Entering a complex contour formula

You can use soft keys to interlink various contours in a mathematical formula.

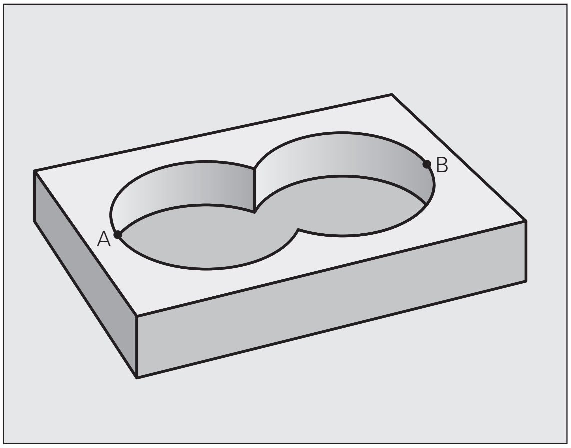
SPEC FCT	▶ Show the soft-key row with special functions
CONTOUR + POINT MACHINING	▶ Select the menu for functions for contour and point machining
COMPLEX CONTOUR FORMULAS	▶ Select the COMPLEX CONTOUR FORMULA menu
CONTOUR FORMULA	▶ Press the CONTOUR FORMULA soft key. The TNC then displays the following soft keys:

Mathematical function	Soft key
Intersected with e.g. `QC10 = QC1 & QC5`	
Joined with e.g. `QC25 = QC7 | QC18`	
Joined without intersection e.g. `QC12 = QC5 ^ QC25`	
Intersected with complement of e.g. `QC25 = QC1 \ QC2`	
Complement of the contour area e.g. `QC12 = #QC11`	
Opening parenthesis e.g. `QC12 = QC1 * (QC2 + QC3)`	(
Closing parenthesis e.g. `QC12 = QC1 * (QC2 + QC3)`	)
Defining a single contour e.g. `QC12 = QC1`	

Overlapping contours

By default, the TNC considers a programmed contour to be a pocket. With the functions of the contour formula, you can convert a contour from a pocket to an island.

Pockets and islands can be overlapped to form a new contour. You can thus enlarge the area of a pocket by another pocket or reduce it by an island.



Subprograms: overlapping pockets

> The following programming examples are contour description programs that are defined in a contour definition program. The contour definition program is called through the **SEL CONTOUR** function in the actual main program.

Pockets A and B overlap.

The TNC calculates the points of intersection S1 and S2 (they do not have to be programmed).

The pockets are programmed as full circles.

Contour description program 1: pocket A

```
0 BEGIN PGM POCKET_A MM
1 L X+10 Y+50 R0
2 CC X+35 Y+50
3 C X+10 Y+50 DR-
4 END PGM POCKET_A MM
```

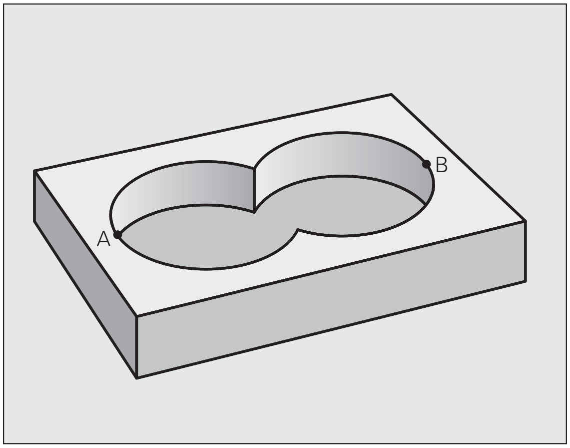
Contour description program 2: pocket B

```
0 BEGIN PGM POCKET_B MM
1 L X+90 Y+50 R0
2 CC X+65 Y+50
3 C X+90 Y+50 DR-
4 END PGM POCKET_B MM
```

Area of inclusion

Both areas A and B are to be machined, including the overlapping area:

- The areas A and B must be entered in separate programs without radius compensation.
- In the contour formula, the areas A and B are processed with the "joined with" function.



Contour definition program:

```
50 ...
51 ...
52 DECLARE CONTOUR QC1 = "POCKET_A.H"
53 DECLARE CONTOUR QC2 = "POCKET_B.H"
54 QC10 = QC1 | QC2
55 ...
56 ...
```

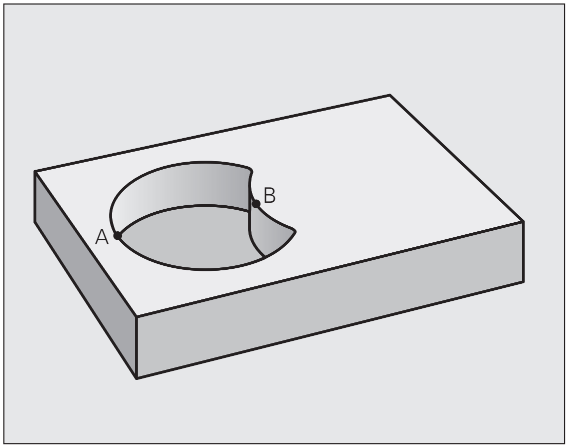
Area of exclusion

Area A is to be machined without the portion overlapped by B:

- The areas A and B must be entered in separate programs without radius compensation.
- In the contour formula, the area B is subtracted from the area A with the "intersected with complement of" function.

Contour definition program:

```
50 ...
51 ...
52 DECLARE CONTOUR QC1 = "POCKET_A.H"
53 DECLARE CONTOUR QC2 = "POCKET_B.H"
54 QC10 = QC1 \ QC2
55 ...
56 ...
```



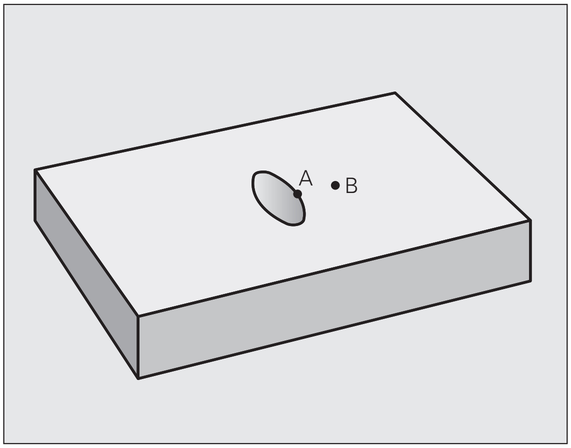
Area of intersection

Only the area where A and B overlap is to be machined. (The areas covered by A or B alone are to be left unmachined.)

- The areas A and B must be entered in separate programs without radius compensation.
- In the contour formula, the areas A and B are processed with the "intersection with" function.

Contour definition program:

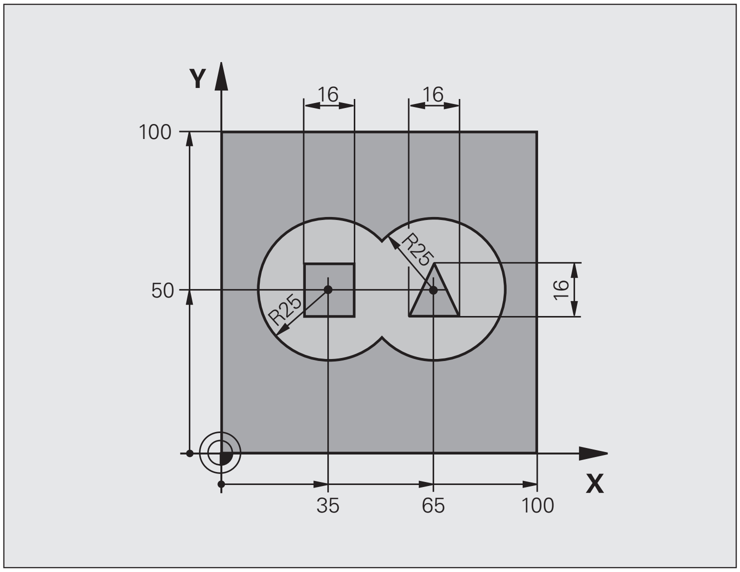
```
50 ...
51 ...
52 DECLARE CONTOUR QC1 = "POCKET_A.H"
53 DECLARE CONTOUR QC2 = "POCKET_B.H"
54 QC10 = QC1 & QC2
55 ...
56 ...
```



Contour machining with SL Cycles

The complete contour is machined with the SL Cycles 20 to 24 (see "Overview" on page 186).

Example: Roughing and finishing superimposed contours with the contour formula



`0 BEGIN PGM CONTOUR MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-40`	Definition of workpiece blank
`2 BLK FORM 0.2 X+100 Y+100 Z+0`	
`3 TOOL DEF 1 L+0 R+2.5`	Tool definition of roughing cutter
`4 TOOL DEF 2 L+0 R+3`	Tool definition of finishing cutter
`5 TOOL CALL 1 Z S2500`	Tool call of roughing cutter
`6 L Z+250 R0 FMAX`	Retract the tool
`7 SEL CONTOUR "MODEL"`	Specify contour definition program
`8 CYCL DEF 20 CONTOUR DATA`	Define general machining parameters
`Q1=-20 ;MILLING DEPTH`	
`Q2=1 ;TOOL PATH OVERLAP`	
`Q3=+0.5 ;ALLOWANCE FOR SIDE`	
`Q4=+0.5 ;ALLOWANCE FOR FLOOR`	
`Q5=+0 ;SURFACE COORDINATE`	
`Q6=2 ;SET-UP CLEARANCE`	
`Q7=+100 ;CLEARANCE HEIGHT`	
`Q8=0.1 ;ROUNDING RADIUS`	
`Q9=-1 ;DIRECTION OF ROTATION`	
`9 CYCL DEF 22 ROUGH-OUT`	Cycle definition: Rough-out
`Q10=5 ;PLUNGING DEPTH`	

Q11=100 ;FEED RATE FOR PLNGNG	
Q12=350 ;FEED RATE FOR ROUGHING	
Q18=0 ;COARSE ROUGHING TOOL	
Q19=150 ;RECIPROCATION FEED RATE	
Q401=100 ;FEED RATE FACTOR	
Q404=0 ;FINE ROUGH STRATEGY	
10 CYCL CALL M3	Cycle call: Rough-out
11 TOOL CALL 2 Z S5000	Tool call of finishing cutter
12 CYCL DEF 23 FLOOR FINISHING	Cycle definition: Floor finishing
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=200 ;FEED RATE FOR ROUGHING	
13 CYCL CALL M3	Cycle call: Floor finishing
14 CYCL DEF 24 SIDE FINISHING	Cycle definition: Side finishing
Q9=+1 ;DIRECTION OF ROTATION	
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=400 ;FEED RATE FOR ROUGHING	
Q14=+0 ;ALLOWANCE FOR SIDE	
15 CYCL CALL M3	Cycle call: Side finishing
16 L Z+250 R0 FMAX M2	Retract the tool, end program
17 END PGM CONTOUR MM	

Contour definition program with contour formula:

0 BEGIN PGM MODEL MM	Contour definition program
1 DECLARE CONTOUR QC1 = "CIRCLE1"	Definition of the contour designator for the program "CIRCLE1"
2 FN 0: Q1 =+35	Assignment of values for parameters used in PGM "CIRCLE31XY"
3 FN 0: Q2 =+50	
4 FN 0: Q3 =+25	
5 DECLARE CONTOUR QC2 = "CIRCLE31XY"	Definition of the contour designator for the program "CIRCLE31XY"
6 DECLARE CONTOUR QC3 = "TRIANGLE"	Definition of the contour designator for the program "TRIANGLE"
7 DECLARE CONTOUR QC4 = "SQUARE"	Definition of the contour designator for the program "SQUARE"
8 QC10 = (QC 1 | QC 2) \ QC 3 \ QC 4	Contour formula
9 END PGM MODEL MM	

Contour description programs:

0 BEGIN PGM CIRCLE1 MM	Contour description program: circle at right
1 CC X+65 Y+50	
2 L PR+25 PA+0 R0	
3 CP IPA+360 DR+	
4 END PGM CIRCLE1 MM	

0 BEGIN PGM CIRCLE31XY MM	Contour description program: circle at left
1 CC X+Q1 Y+Q2	
2 LP PR+Q3 PA+0 R0	
3 CP IPA+360 DR+	
4 END PGM CIRCLE31XY MM	

0 BEGIN PGM TRIANGLE MM	Contour description program: triangle at right
1 L X+73 Y+42 R0	
2 L X+65 Y+58	
3 L X+58 Y+42	
4 L X+73	
5 END PGM TRIANGLE MM	

0 BEGIN PGM SQUARE MM	Contour description program: square at left
1 L X+27 Y+58 R0	
2 L X+43	
3 L Y+42	
4 L X+27	
5 L Y+58	
6 END PGM SQUARE MM	

9.2 SL cycles with simple contour formula

Fundamentals

SL cycles and the simple contour formula enable you to form contours by combining up to 9 subcontours (pockets or islands) in a simple manner. You define the individual subcontours (geometry data) as separate programs. In this way, any subcontour can be used any number of times. The TNC calculates the contour from the selected subcontours.

The memory capacity for programming an SL cycle (all contour description programs) is limited to **128 contours.** The number of possible contour elements depends on the type of contour (inside or outside contour) and the number of contour descriptions. You can program up to approx. **8192** elements.

Properties of the subcontours

- By default, the TNC assumes that the contour is a pocket. Do not program a radius compensation.
- The TNC ignores feed rates F and miscellaneous functions M.
- Coordinate transformations are allowed. If they are programmed within the subcontour they are also effective in the following subprograms, but they need not be reset after the cycle call.
- Although the subprograms can contain coordinates in the spindle axis, such coordinates are ignored.
- The working plane is defined in the first coordinate block of the subprogram. The secondary axes U,V,W are permitted.

Example: Program structure: Machining with SL cycles and complex contour formula

```
0 BEGIN PGM CONTDEF MM
...
5 CONTOUR DEF
P1= "POCK1.H"
I2 = "ISLE2.H" DEPTH5
I3 "ISLE3.H" DEPTH7.5
6 CYCL DEF 20 CONTOUR DATA ...
8 CYCL DEF 22 ROUGH-OUT ...
9 CYCL CALL
...
12 CYCL DEF 23 FLOOR FINISHING ...
13 CYCL CALL
...
16 CYCL DEF 24 SIDE FINISHING ...
17 CYCL CALL
63 L Z+250 R0 FMAX M2
64 END PGM CONTDEF MM
```

Characteristics of the fixed cycles

- The TNC automatically positions the tool to the set-up clearance before a cycle.
- Each level of infeed depth is milled without interruptions since the cutter traverses around islands instead of over them.
- The radius of "inside corners" can be programmed—the tool keeps moving to prevent surface blemishes at inside corners (this applies to the outermost pass in the Rough-out and Side Finishing cycles).
- The contour is approached in a tangential arc for side finishing.
- For floor finishing, the tool again approaches the workpiece on a tangential arc (for tool axis Z, for example, the arc is in the Z/X plane).
- The contour is machined throughout in either climb or up-cut milling.

With Machine Parameter 7420 you can determine where the tool is positioned at the end of Cycles 21 to 24.

The machining data (such as milling depth, finishing allowance and set-up clearance) are entered as CONTOUR DATA in Cycle 20.

Entering a simple contour formula

You can use soft keys to interlink various contours in a mathematical formula.

SPEC FCT

- Show the soft-key row with special functions

CONTOUR + POINT MACHINING

- Select the menu for functions for contour and point machining

CONTOUR DEF

- Press the CONTOUR DEF soft key. The TNC opens the dialog for entering the contour formula
- Select the name of the first subcontour with the WINDOW SELECTION soft key or enter it directly The first subcontour must always be the deepest pocket. Confirm with the ENT key

ISLAND

- Specify via soft key whether the next subcontour is a pocket or an island. Confirm with the ENT key
- Select the name of the second subcontour with the WINDOW SELECTION soft key or enter it directly. Confirm by pressing the ENT key
- If needed, enter the depth of the second subcontour. Confirm with the ENT key
- Carry on with the dialog as described above until you have entered all subcontours.

- Always start the list of subcontours with the deepest pocket!
- If the contour is defined as an island, the TNC interprets the entered depth as the island height. The entered value (without an algebraic sign) then refers to the workpiece top surface!
- If the depth is entered as 0, then for pockets the depth defined in the Cycle 20 is effective. Islands then rise up to the workpiece top surface!

Contour machining with SL Cycles

The complete contour is machined with the SL Cycles 20 to 24 (see "Overview" on page 186).

10

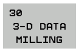
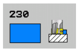
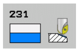
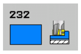
Fixed Cycles: Multipass Milling

10.1 Fundamentals

Overview

The TNC offers four cycles for machining surfaces with the following characteristics:

- Created with a CAD/CAM system
- Flat, rectangular surfaces
- Flat, oblique-angled surfaces
- Surfaces that are inclined in any way
- Twisted surfaces

Cycle	Soft key	Page
30 RUN 3-D DATA For multipass milling of 3-D data in several infeeds		Page 263
230 MULTIPASS MILLING For flat rectangular surfaces		Page 265
231 RULED SURFACE For oblique, inclined or twisted surfaces		Page 267
232 FACE MILLING For level rectangular surfaces, with indicated oversizes and multiple infeeds		Page 271

10.2 RUN 3-D DATA (Cycle 30, DIN/ISO: G60)

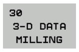
Cycle run

1 From the current position, the TNC positions the tool at rapid traverse **FMAX** in the tool axis to the set-up clearance above the MAX point that you have programmed in the cycle.
2 The tool then moves at **FMAX** in the working plane to the MIN point you have programmed in the cycle.
3 From this point, the tool advances to the first contour point at the feed rate for plunging.
4 The TNC subsequently processes all points that are stored in the digitizing data file at the **feed rate for milling.** If necessary, the TNC retracts the tool between machining operations to the **set-up clearance** if specific areas are to be left unmachined.
5 At the end of the cycle, the tool is retracted at **FMAX** to the set-up clearance.

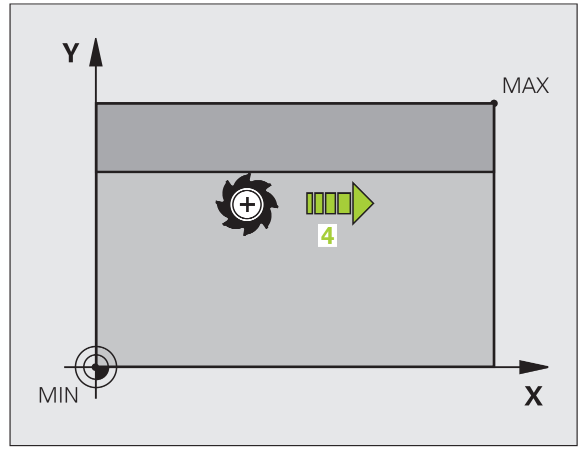
Please note while programming:

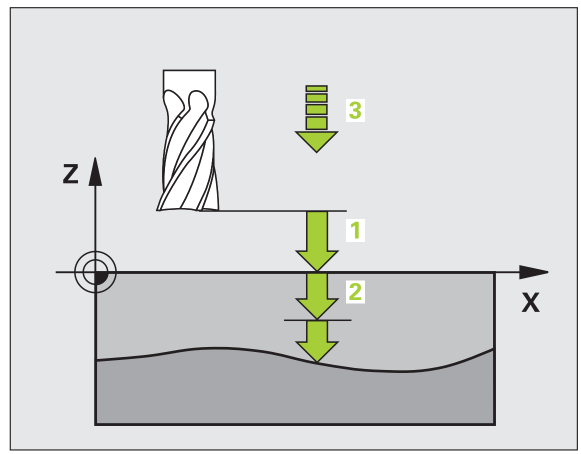
You can particularly use Cycle 30 to run conversational programs created offline in multiple infeeds.

Cycle parameters



- **PGM name 3-D data**: Enter the name of the program in which the contour data is stored. If the file is not stored in the current directory, enter the complete path. A maximum of 254 characters can be entered.
- **Min. point of range**: Lowest coordinates (X, Y and Z coordinates) in the range to be milled. Input range -99999.9999 to 99999.9999
- **Max. point of range**: Largest coordinates (X, Y and Z coordinates) in the range to be milled. Input range -99999.9999 to 99999.9999
- **Set-up clearance** 1 (incremental): Distance between tool tip and workpiece surface for tool movements at rapid traverse. Input range 0 to 99999.9999
- **Plunging depth** 2 (incremental): Infeed per cut Input range -99999.9999 to 99999.9999
- **Feed rate for plunging** 3: Traversing speed of the tool in mm/min when moving into the workpiece. Input range 0 to 99999.999; alternatively **FAUTO**
- **Feed rate for milling** 4: Traversing speed of the tool in mm/min while milling. Input range 0 to 99999.9999; alternatively **FAUTO**
- **Miscellaneous function M**: Optional entry of one to two miscellaneous functions, for example M13. Input range 0 to 999



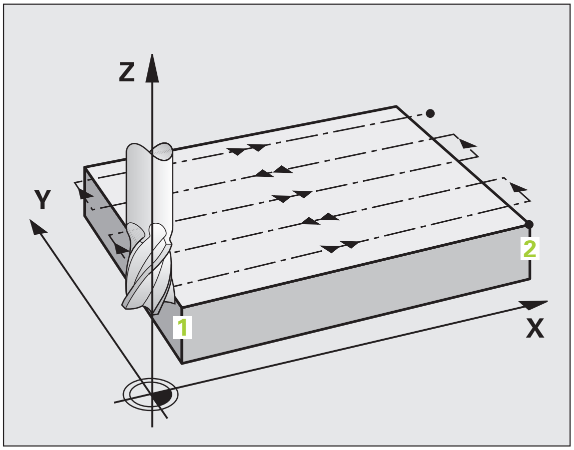


Example: NC blocks

```
64 CYCL DEF 30.0 RUN 3-D DATA
65 CYCL DEF 30.1 PGM DIGIT.: BSP.H
66 CYCL DEF 30.2 X+0 Y+0 Z-20
67 CYCL DEF 30.3 X+100 Y+100 Z+0
68 CYCL DEF 30.4 SETUP 2
69 CYCL DEF 30.5 PECKG -5 F100
70 CYCL DEF 30.6 F350 M8
```

10.3 MULTIPASS MILLING (Cycle 230, DIN/ISO: G230)

Cycle run



1 From the current position in the working plane, the TNC positions the tool at rapid traverse **FMAX** to the starting point **1**; the TNC moves the tool by its radius to the left and upward.
2 The tool then moves at **FMAX** in the tool axis to the set-up clearance. From there it approaches the programmed starting position in the tool axis at the feed rate for plunging.
3 The tool then moves at the programmed feed rate for milling to the end point **2**. The TNC calculates the end point from the programmed starting point, the programmed length, and the tool radius.
4 The TNC offsets the tool to the starting point in the next pass at the stepover feed rate. The offset is calculated from the programmed width and the number of cuts.
5 The tool then returns in the negative direction of the first axis.
6 Multipass milling is repeated until the programmed surface has been completed.
7 At the end of the cycle, the tool is retracted at **FMAX** to the set-up clearance.

Please note while programming:

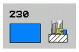
From the current position, the TNC positions the tool at the starting point, first in the working plane and then in the spindle axis.

Pre-position the tool in such a way that no collision with the workpiece or the fixtures can occur.

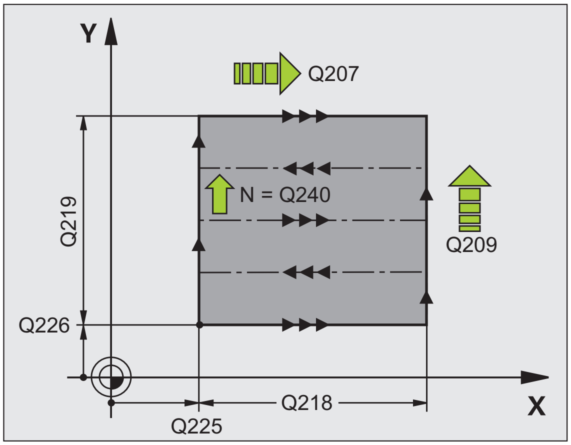
Danger of collision!

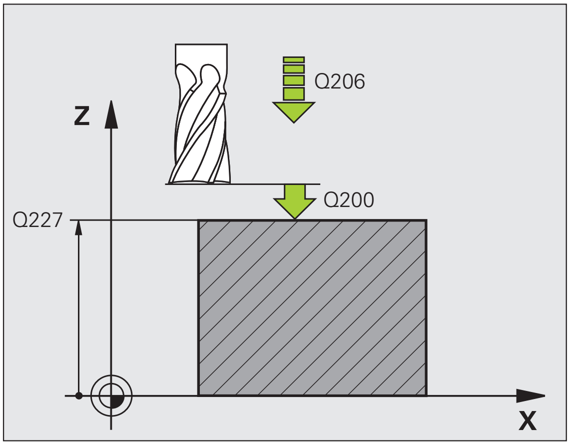
Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if spindle rotation is not active when the cycle is called. The function also needs to be adapted by your machine manufacturer.

Cycle parameters



- **Starting point in 1st axis** Q225 (absolute): Minimum-point coordinate of the surface to be multipass-milled in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Starting point in 2nd axis** Q226 (absolute): Minimum-point coordinate of the surface to be multipass-milled in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Starting point in 3rd axis** Q227 (absolute): Height in the spindle axis at which multipass-milling is carried out. Input range -99999.9999 to 99999.9999
- **First side length** Q218 (incremental): Length of the surface to be multipass-milled in the reference axis of the working plane, referenced to the starting point in the 1st axis. Input range 0 to 99999.9999
- **Second side length** Q219 (incremental): Length of the surface to be multipass-milled in the minor axis of the working plane, referenced to the starting point in the 2nd axis. Input range 0 to 99999.9999
- **Number of cuts** Q240: Number of passes to be made over the width. Input range 0 to 99999
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min when moving from set-up clearance to the milling depth. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Stepover feed rate** Q209: Traversing speed of the tool in mm/min when moving to the next pass. If you are moving the tool transversely in the material, enter Q209 to be smaller than Q207. If you are moving it transversely in the open, Q209 may be greater than Q207. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Set-up clearance** Q200 (incremental): Distance between tool tip and milling depth for positioning at the start and end of the cycle. Input range 0 to 99999.9999; alternatively **PREDEF**



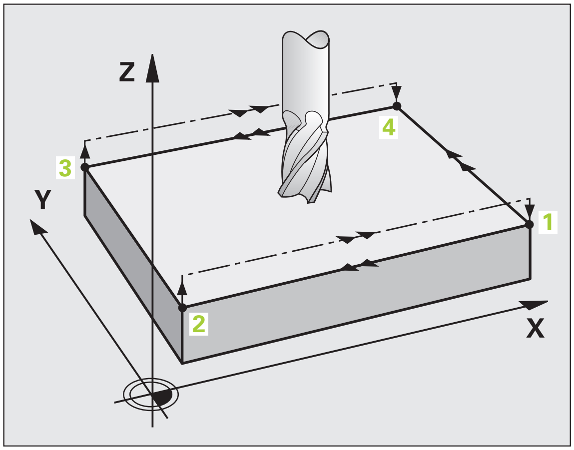


Example: NC blocks

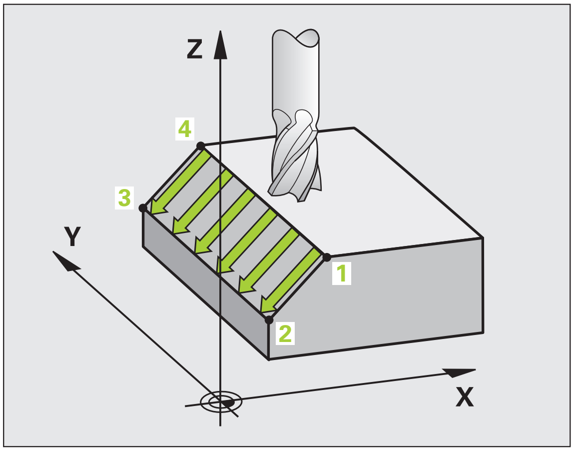
```
71 CYCL DEF 230 MULTIPASS MILLING
    Q225=+10  ;STARTING POINT 1ST AXIS
    Q226=+12  ;STARTING POINT 2ND AXIS
    Q227=+2.5 ;STARTING POINT 3RD AXIS
    Q218=150  ;1ST SIDE LENGTH
    Q219=75   ;2ND SIDE LENGTH
    Q240=25   ;NUMBER OF CUTS
    Q206=150  ;FEED RATE FOR PLNGNG
    Q207=500  ;FEED RATE FOR MILLING
    Q209=200  ;STEPOVER FEED RATE
    Q200=2    ;SET-UP CLEARANCE
```

10.4 RULED SURFACE (Cycle 231, DIN/ISO: G231)

Cycle run

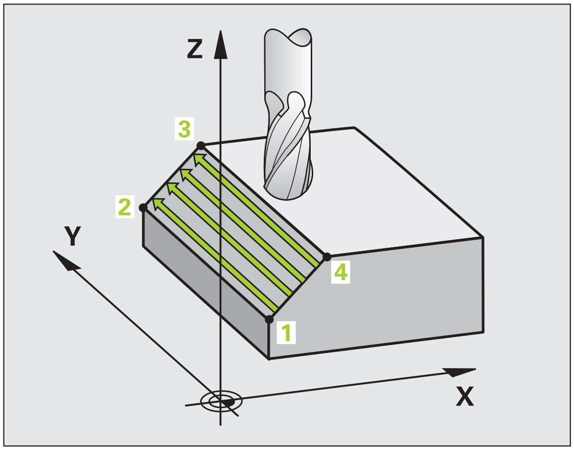


1. From the current position, the TNC positions the tool in a linear 3-D movement to the starting point **1**.
2. The tool subsequently advances to the end point **2** at the programmed feed rate for milling.
3. From this point, the tool moves at rapid traverse **FMAX** by the tool diameter in the positive tool axis direction, and then back to starting point **1**.
4. At the starting point **1** the TNC moves the tool back to the last traversed Z value.
5. Then the TNC moves the tool in all three axes from point **1** in the direction of point **4** to the next line.
6. From this point, the tool moves to the end point on this pass. The TNC calculates the end point from point **2** and a movement in the direction of point **3**.
7. Multipass milling is repeated until the programmed surface has been completed.
8. At the end of the cycle, the tool is positioned above the highest programmed point in the spindle axis, offset by the tool diameter.



Cutting motion

The starting point, and therefore the milling direction, is selectable because the TNC always moves from point **1** to point **2** and in the total movement from point **1** / **2** to point **3** / **4**. You can program point **1** at any corner of the surface to be machined.



If you are using an end mill for the machining operation, you can optimize the surface finish in the following ways:

- A shaping cut (spindle-axis coordinate of point **1** greater than spindle-axis coordinate of point **2**) for slightly inclined surfaces.
- A drawing cut (spindle-axis coordinate of point **1** smaller than spindle-axis coordinate of point **2**) for steep surfaces.
- When milling twisted surfaces, program the main cutting direction (from point **1** to point **2**) parallel to the direction of the steeper inclination.

If you are using a spherical cutter for the machining operation, you can optimize the surface finish in the following way:

- When milling twisted surfaces, program the main cutting direction (from point **1** to point **2**) perpendicular to the direction of the steepest inclination.

Please note while programming:

From the current position, the TNC positions the tool in a linear 3-D movement to the starting point **1**. Pre-position the tool in such a way that no collision with the workpiece or the fixtures can occur.

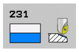
The TNC moves the tool with radius compensation R0 to the programmed positions.

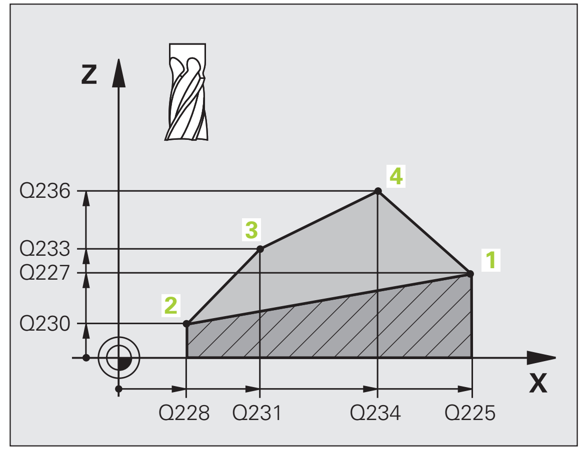
If required, use a center-cut end mill (ISO 1641).

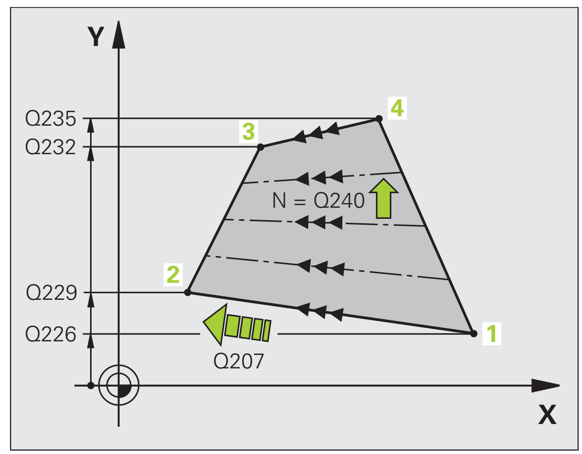
Danger of collision!

Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if spindle rotation is not active when the cycle is called. The function also needs to be adapted by your machine manufacturer.

Cycle parameters







- **`Starting point in 1st axis`** Q225 (absolute): Starting point coordinate of the surface to be multipass-milled in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **`Starting point in 2nd axis`** Q226 (absolute): Starting point coordinate of the surface to be multipass-milled in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **`Starting point in 3rd axis`** Q227 (absolute): Starting point coordinate of the surface to be multipass-milled in the tool axis. Input range -99999.9999 to 99999.9999
- **`2nd point in 1st axis`** Q228 (absolute): End point coordinate of the surface to be multipass-milled in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **`2nd point in 2nd axis`** Q229 (absolute): End point coordinate of the surface to be multipass-milled in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **`2nd point in 3rd axis`** Q230 (absolute): End point coordinate of the surface to be multipass-milled in the spindle axis. Input range -99999.9999 to 99999.9999
- **`3rd point in 1st axis`** Q231 (absolute): Coordinate of point **3** in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **`3rd point in 2nd axis`** Q232 (absolute): Coordinate of point **3** in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **`3rd point in 3rd axis`** Q233 (absolute): Coordinate of point **3** in the spindle axis. Input range -99999.9999 to 99999.9999

- **4th point in 1st axis** Q234 (absolute): Coordinate of point **4** in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **4th point in 2nd axis** Q235 (absolute): Coordinate of point **4** in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **4th point in 3rd axis** Q236 (absolute): Coordinate of point **4** in the spindle axis. Input range -99999.9999 to 99999.9999
- **Number of cuts** Q240: Number of passes to be made between points **1** and **4**, **2** and **3**. Input range 0 to 99999
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling. The TNC performs the first step at half the programmed feed rate. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**

Example: NC blocks

```
72 CYCL DEF 231 RULED SURFACE
    Q225=+0   ;STARTING POINT 1ST AXIS
    Q226=+5   ;STARTING POINT 2ND AXIS
    Q227=-2   ;STARTING POINT 3RD AXIS
    Q228=+100 ;2ND POINT 1ST AXIS
    Q229=+15  ;2ND POINT 2ND AXIS
    Q230=+5   ;2ND POINT 3RD AXIS
    Q231=+15  ;3RD POINT 1ST AXIS
    Q232=+125 ;3RD POINT 2ND AXIS
    Q233=+25  ;3RD POINT 3RD AXIS
    Q234=+15  ;4TH POINT 1ST AXIS
    Q235=+125 ;4TH POINT 2ND AXIS
    Q236=+25  ;4TH POINT 3RD AXIS
    Q240=40   ;NUMBER OF CUTS
    Q207=500  ;FEED RATE FOR MILLING
```

10.5 FACE MILLING (Cycle 232, DIN/ISO: G232)

Cycle run

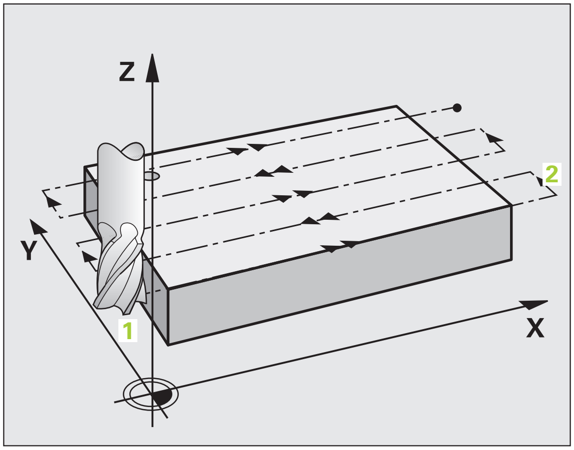
Cycle 232 is used to face mill a level surface in multiple infeeds while taking the finishing allowance into account. Three machining strategies are available:

- **Strategy Q389=0**: Meander machining, stepover outside the surface being machined
- **Strategy Q389=1**: Meander machining, stepover within the surface being machined
- **Strategy Q389=2**: Line-by-line machining, retraction and stepover at the positioning feed rate

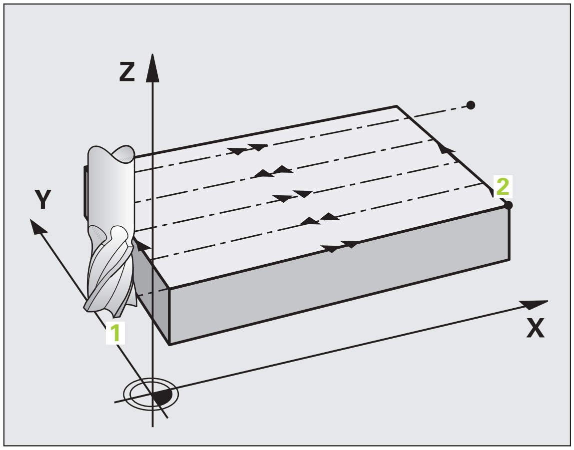
1 From the current position, the TNC positions the tool at rapid traverse **FMAX** to the starting point 1 using positioning logic: If the current position in the spindle axis is greater than the 2nd set-up clearance, the TNC positions the tool first in the machining plane and then in the spindle axis. Otherwise it first moves to the 2nd set-up clearance and then in the machining plane. The starting point in the machining plane is offset from the edge of the workpiece by the tool radius and the safety clearance to the side.
2 The tool then moves in the spindle axis at the positioning feed rate to the first plunging depth calculated by the TNC.

Strategy Q389=0

3 The tool then advances to the end point 2 at the programmed feed rate for milling. The end point lies **outside** the surface. The TNC calculates the end point from the programmed starting point, the programmed length, the programmed safety clearance to the side and the tool radius.
4 The TNC offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius and the maximum path overlap factor.
5 The tool then moves back in the direction of the starting point 1.
6 The process is repeated until the programmed surface has been completed. At the end of the last pass, the tool plunges to the next machining depth.
7 In order to avoid non-productive motions, the surface is then machined in reverse direction.
8 The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.
9 At the end of the cycle, the tool is retracted at **FMAX** to the 2nd set-up clearance.



Strategy Q389=1



3 The tool then advances to the end point **2** at the programmed feed rate for milling. The end point lies **within** the surface. The TNC calculates the end point from the programmed starting point, the programmed length and the tool radius.

4 The TNC offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius and the maximum path overlap factor.

5 The tool then moves back in the direction of the starting point **1**. The motion to the next line occurs within the workpiece borders.

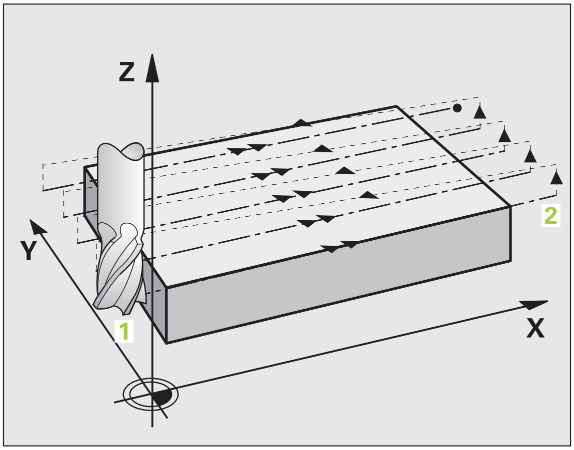
6 The process is repeated until the programmed surface has been completed. At the end of the last pass, the tool plunges to the next machining depth.

7 In order to avoid non-productive motions, the surface is then machined in reverse direction.

8 The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.

9 At the end of the cycle, the tool is retracted at `FMAX` to the 2nd set-up clearance.

Strategy Q389=2



3 The tool then advances to the end point **2** at the programmed feed rate for milling. The end point lies outside the surface. The control calculates the end point from the programmed starting point, the programmed length, the programmed safety clearance to the side and the tool radius.

4 The TNC positions the tool in the spindle axis to the set-up clearance over the current infeed depth, and then moves at the pre-positioning feed rate directly back to the starting point in the next line. The TNC calculates the offset from the programmed width, the tool radius and the maximum path overlap factor.

5 The tool then returns to the current infeed depth and moves in the direction of the next end point **2**.

6 The milling process is repeated until the programmed surface has been completed. At the end of the last pass, the tool plunges to the next machining depth.

7 In order to avoid non-productive motions, the surface is then machined in reverse direction.

8 The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.

9 At the end of the cycle, the tool is retracted at `FMAX` to the 2nd set-up clearance.

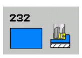
Please note while programming:

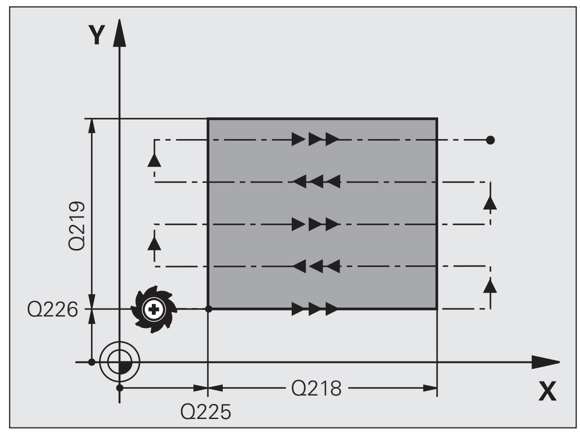
Enter the 2nd set-up clearance in Q204 so that no collision with the workpiece or the fixtures can occur.

Danger of collision!

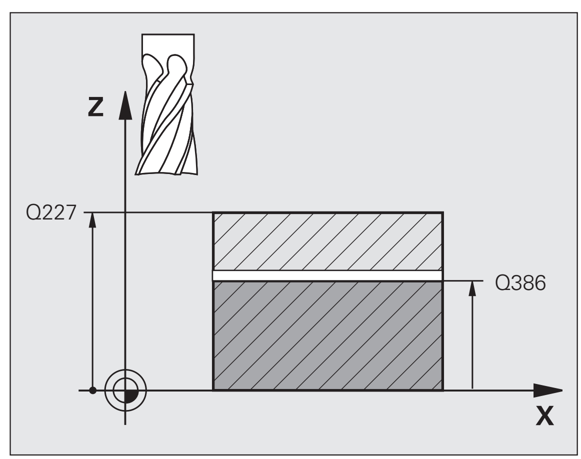
Enter in MP7441 bit 0 whether the TNC should output an error message (bit 0=0) or not (bit 0=1) if spindle rotation is not active when the cycle is called. The function also needs to be adapted by your machine manufacturer.

Cycle parameters

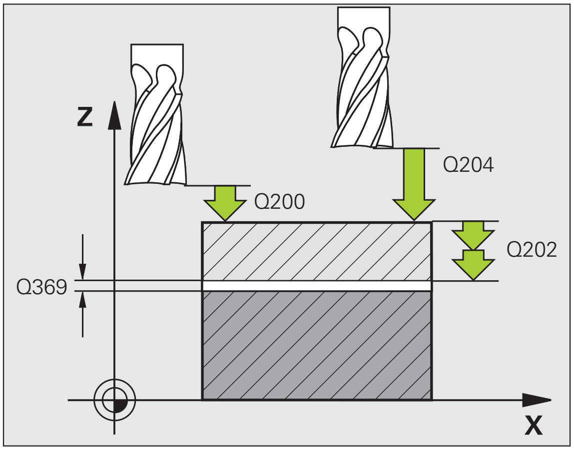


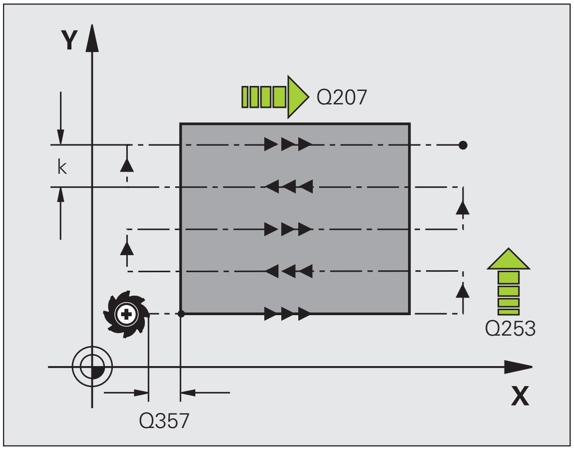


- **Machining strategy (0/1/2)** Q389: Specify how the TNC is to machine the surface:
  0: Meander machining, stepover at positioning feed rate outside the surface to be machined
  1: Meander machining, stepover at feed rate for milling within the surface to be machined
  2: Line-by-line machining, retraction and stepover at the positioning feed rate
- **Starting point in 1st axis** Q225 (absolute): Starting point coordinate of the surface to be machined in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Starting point in 2nd axis** Q226 (absolute): Starting point coordinate of the surface to be multipass-milled in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Starting point in 3rd axis** Q227 (absolute): Coordinate of the workpiece surface used to calculate the infeeds. Input range -99999.9999 to 99999.9999
- **End point in 3rd axis** Q386 (absolute): Coordinate in the spindle axis to which the surface is to be face milled. Input range -99999.9999 to 99999.9999



- **`1st side length`** Q218 (incremental): Length of the surface to be machined in the reference axis of the working plane. Use the algebraic sign to specify the direction of the first milling path in reference to the **`starting point in the 1st axis`**. Input range -99999.9999 to 99999.9999
- **`2nd side length`** Q219 (incremental): Length of the surface to be machined in the minor axis of the working plane. Use the algebraic sign to specify the direction of the first stepover in reference to the **`starting point in the 2nd axis`**. Input range -99999.9999 to 99999.9999
- **`Maximum plunging depth`** Q202 (incremental): **Maximum** amount that the tool is advanced each time. The TNC calculates the actual plunging depth from the difference between the end point and starting point of the tool axis (taking the finishing allowance into account), so that uniform plunging depths are used each time. Input range 0 to 99999.9999
- **`Allowance for floor`** Q369 (incremental): Distance used for the last infeed. Input range 0 to 99999.9999
- **`Max. path overlap factor`** Q370: **Maximum** stepover factor k. The TNC calculates the actual stepover from the second side length (Q219) and the tool radius so that a constant stepover is used for machining. If you have entered a radius R2 in the tool table (e.g. tooth radius when using a face-milling cutter), the TNC reduces the stepover accordingly. Input range 0.1 to 1.9999; alternatively **`PREDEF`**





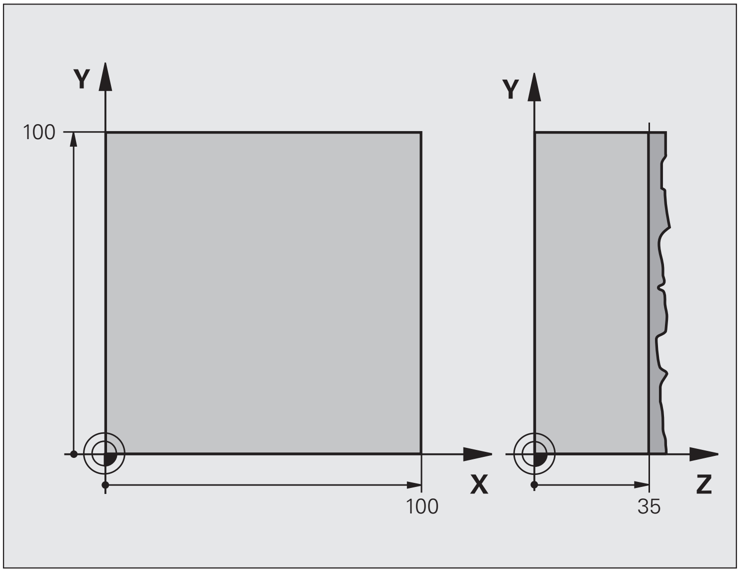
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for finishing** Q385: Traversing speed of the tool in mm/min while milling the last infeed. Input range 0 to 99999.9999; alternatively **FAUTO**, **FU**, **FZ**
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool in mm/min when approaching the starting position and when moving to the next pass. If you are moving the tool transversely to the material (Q389=1), the TNC moves the tool at the feed rate for milling Q207. Input range 0 to 99999.9999; alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Set-up clearance** Q200 (incremental): Distance between tool tip and the starting position in the tool axis. If you are milling with machining strategy Q389=2, the TNC moves the tool at the set-up clearance over the current plunging depth to the starting point of the next pass. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Clearance to side** Q357 (incremental): Safety clearance to the side of the workpiece when the tool approaches the first plunging depth, and distance at which the stepover occurs if the machining strategy Q389=0 or Q389=2 is used. Input range 0 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**

Example: NC blocks

```
71 CYCL DEF 232 FACE MILLING
    Q389=2     ;STRATEGY
    Q225=+10   ;STARTING POINT 1ST AXIS
    Q226=+12   ;STARTING POINT 2ND AXIS
    Q227=+2.5  ;STARTING POINT 3RD AXIS
    Q386=-3    ;END POINT IN 3RD AXIS
    Q218=150   ;1ST SIDE LENGTH
    Q219=75    ;2ND SIDE LENGTH
    Q202=2     ;MAX. PLUNGING DEPTH
    Q369=0.5   ;ALLOWANCE FOR FLOOR
    Q370=1     ;MAX. OVERLAP
    Q207=500   ;FEED RATE FOR MILLING
    Q385=800   ;FEED RATE FOR FINISHING
    Q253=2000  ;F PRE-POSITIONING
    Q200=2     ;SET-UP CLEARANCE
    Q357=2     ;CLEARANCE TO SIDE
    Q204=2     ;2ND SET-UP CLEARANCE
```

10.6 Programming examples

Example: Multipass milling



0 BEGIN PGM C230 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z+0	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+40	
3 TOOL DEF 1 L+0 R+5	Tool definition
4 TOOL CALL 1 Z S3500	Tool call
5 L Z+250 R0 FMAX	Retract the tool
6 CYCL DEF 230 MULTIPASS MILLING	Cycle definition: MULTIPASS MILLING
Q225=+0 ;STARTING POINT 1ST AXIS	
Q226=+0 ;STARTING POINT 2ND AXIS	
Q227=+35 ;STARTING POINT 3RD AXIS	
Q218=100 ;1ST SIDE LENGTH	
Q219=100 ;2ND SIDE LENGTH	
Q240=25 ;NUMBER OF CUTS	
Q206=250 ;FEED RATE FOR PLNGNG	
Q207=400 ;FEED RATE FOR MILLING	
Q209=150 ;STEPOVER FEED RATE	
Q200=2 ;SET-UP CLEARANCE	

`7 L X+-25 Y+0 R0 FMAX M3`	Pre-position near the starting point
`8 CYCL CALL`	Cycle call
`9 L Z+250 R0 FMAX M2`	Retract the tool, end program
`10 END PGM C230 MM`	

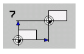
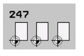
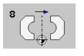
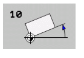
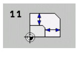
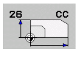
11

Cycles: Coordinate Transformations

11.1 Fundamentals

Overview

Once a contour has been programmed, you can position it on the workpiece at various locations and in different sizes through the use of coordinate transformations. The TNC provides the following coordinate transformation cycles:

Cycle	Soft key	Page
7 DATUM SHIFT For shifting contours directly within the program or from datum tables		Page 281
247 DATUM SETTING Datum setting during program run		Page 288
8 MIRROR IMAGE Mirroring contours		Page 289
10 ROTATION Rotating contours in the working plane		Page 291
11 SCALING For increasing or reducing the size of contours		Page 293
26 AXIS-SPECIFIC SCALING FACTOR For increasing or reducing the size of contours with scaling factors for each axis		Page 295
19 WORKING PLANE Machining in tilted coordinate system on machines with swivel heads and/or rotary tables		Page 297

Effect of coordinate transformations

Beginning of effect: A coordinate transformation becomes effective as soon as it is defined—it is not called separately. It remains in effect until it is changed or canceled.

To cancel coordinate transformations:

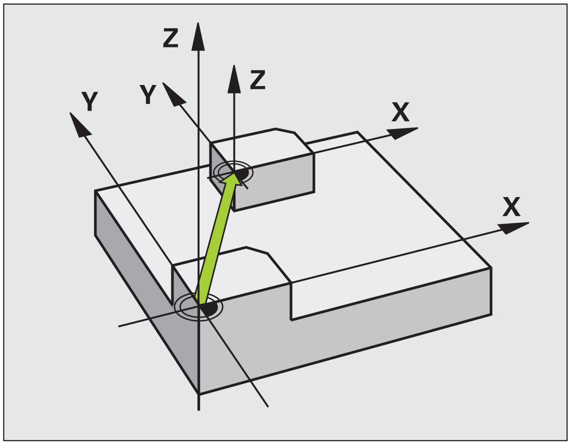
- Define cycles for basic behavior with a new value, such as scaling factor 1.0
- Execute a miscellaneous function M2, M30, or an END PGM block (depending on MP7300)
- Select a new program
- Program miscellaneous function M142 "Erasing modal program information"

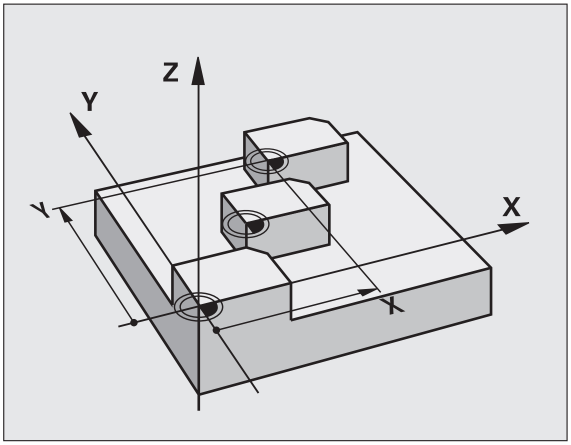
11.2 DATUM SHIFT (Cycle 7, DIN/ISO: G54)

Effect

A DATUM SHIFT allows machining operations to be repeated at various locations on the workpiece.

When the DATUM SHIFT cycle is defined, all coordinate data is based on the new datum. The TNC displays the datum shift in each axis in the additional status display. Input of rotary axes is also permitted.





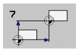
Reset

- Program a datum shift to the coordinates X=0, Y=0 etc. directly with a cycle definition.
- Use the **TRANS DATUM RESET** function.
- Call a datum shift to the coordinates X=0; Y=0 etc. from the datum table.

Graphics

If you program a new **BLK FORM** after a datum shift, you can use MP7310 to determine whether the **BLK FORM** is referenced to the current datum or to the original datum. Referencing a new BLK FORM to the current datum enables you to display each part in a program in which several pallets are machined.

Cycle parameters



- **Datum shift**: Enter the coordinates of the new datum. Absolute values are referenced to the manually set workpiece datum. Incremental values are always referenced to the datum which was last valid—this can be a datum which has already been shifted. Input range: Up to six NC axes, each from –99999.9999 to 99999.9999

Example: NC blocks

```
13 CYCL DEF 7.0 DATUM SHIFT
14 CYCL DEF 7.1 X+60
16 CYCL DEF 7.3 Z-5
15 CYCL DEF 7.2 Y+40
```

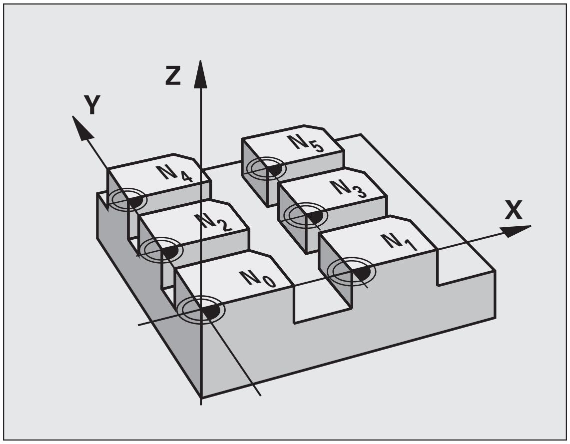
11.3 DATUM SHIFT with datum tables (Cycle 7, DIN/ISO: G53)

Effect

Datum tables are used for:

- Frequently recurring machining sequences at various locations on the workpiece
- Frequent use of the same datum shift

Within a program, you can either program datum points directly in the cycle definition or call them from a datum table.

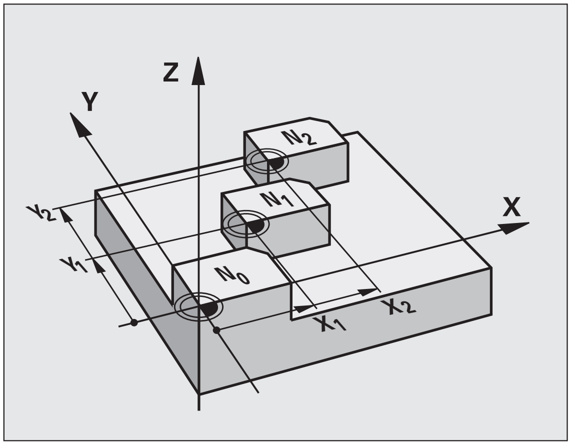


Reset

- Call a datum shift to the coordinates X=0; Y=0 etc. from the datum table.
- Execute a datum shift to the coordinates X=0, Y=0 etc. directly with a cycle definition.
- Use the **TRANS DATUM RESET** function.

Graphic

If you program a new **BLK FORM** after a datum shift, you can use MP7310 to determine whether the **BLK FORM** is referenced to the current datum or to the original datum. Referencing a new BLK FORM to the current datum enables you to display each part in a program in which several pallets are machined.



Status displays

In the additional status display, the following data from the datum table are shown:

- Name and path of the active datum table
- Active datum number
- Comment from the DOC column of the active datum number

Please note while programming:

Danger of collision!

Datums from a datum table are **always and exclusively** referenced to the current datum (preset).

MP7475, which earlier defined whether datums are referenced to the machine datum or the workpiece datum, now serves only as a safety measure. If MP7475 = 1, the TNC outputs an error message if a datum shift is called from a datum table.

Datum tables from the TNC 4xx whose coordinates are referenced to the machine datum (MP7475 = 1) cannot be used in the iTNC 530.

If you are using datum shifts with datum tables, then use the **SEL TABLE** function to activate the desired datum table from the NC program.

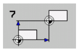
If you work without **SEL TABLE,** then you must activate the desired datum table before the test run or the program run. (This applies also to the programming graphics).

- Use the file management to select the desired table for a test run in the **Test Run** operating mode: The table receives the status S.
- Use the file management in a program run mode to select the desired table for program run: The table receives the status M.

The coordinate values from datum tables are only effective with absolute coordinate values.

New lines can only be inserted at the end of the table.

Cycle parameters



▶ **`Datum shift`**: Enter the number of the datum from the datum table or a Q parameter. If you enter a Q parameter, the TNC activates the datum number entered in the Q parameter. Input range 0 to 9999

Example: NC blocks

```
77 CYCL DEF 7.0 DATUM SHIFT
78 CYCL DEF 7.1 #5
```

Selecting a datum table in the part program

With the **`SEL TABLE`** function you select the table from which the TNC takes the datums:

PGM CALL

▶ Select the functions for program call: Press the PGM CALL key

DATUM TABLE

▶ Press the DATUM TABLE soft key

SELECTION WINDOW

▶ Press the WINDOW SELECTION soft key: The TNC superimposes a window where you can select the desired datum table

▶ Select the desired datum table with the arrow keys or by mouse click and confirm by pressing ENT: The TNC enters the complete path name in the **`SEL TABLE`** block

▶ Conclude this function with the END key

Alternatively you can also enter the table name or the complete path name of the table to be called directly via the keyboard.

Program a **`SEL TABLE`** block before Cycle 7 Datum Shift.

A datum table selected with **`SEL TABLE`** remains active until you select another datum table with **`SEL TABLE`** or through PGM MGT.

You can define datum tables and datum numbers in an NC block with the **`TRANS DATUM TABLE`** function (see Conversational Programming User's Manual).

Editing the datum table in the Programming and Editing mode of operation

After you have changed a value in a datum table, you must save the change with the ENT key. Otherwise the change might not be included during program run.

Select the datum table in the **Programming and Editing** mode of operation.

PGM MGT

- Call the file manager: Press the PGM MGT key
- Display the datum tables: Press the SELECT TYPE and SHOW .D soft keys
- Select the desired table or enter a new file name
- Edit the file The soft-key row comprises the following functions for editing:

Function	Soft key
Select beginning of table	BEGIN
Select end of table	END
Go to previous page	PAGE ↑
Go to next page	PAGE ↓
Insert line (only possible at the end of table)	INSERT LINE
Delete line	DELETE LINE
Confirm the entered line and go to the beginning of the next line	NEXT LINE
Add the entered number of lines (datums) to the end of the table	APPEND N LINES

Editing a datum table in a Program Run operating mode

In a program run mode you can select the active datum table. Press the DATUM TABLE soft key. You can then use the same editing functions as in the **Programming and Editing** mode of operation.

Transferring the actual values into the datum table

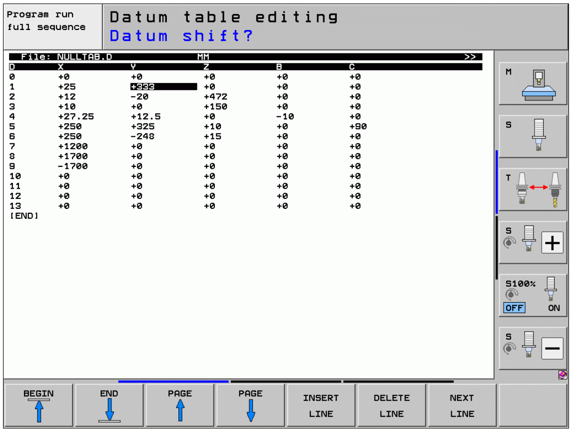
You can enter the current tool position or the last probed position in the datum table by pressing the "actual-position-capture" key:

- Place the text box on the line of the column in which you want to enter the position
- Select the actual-position-capture function: The TNC opens a pop-up window that asks whether you want to enter the current tool position or the last probed values
- Select the desired function with the arrow keys and confirm your selection with the ENT key
- ALL VALUES: To enter the values in all axes, press the ALL VALUES soft key
- PRESENT VALUE: To enter the value in the axis where the text box is located, press the CURRENT VALUE soft key

Configuring the datum table

In the second and third soft-key rows you can define for each datum table the axes for which you wish to set the datums. In the standard setting all of the axes are active. If you wish to exclude an axis, set the corresponding soft key to OFF. The TNC then deletes that column from the datum table.

If you do not wish to define a datum for an active axis, press the NO ENT key. The TNC then enters a dash in that column.



To exit a datum table

Select a different type of file in file management and choose the desired file.

11.4 DATUM SETTING (Cycle 247, DIN/ISO: G247)

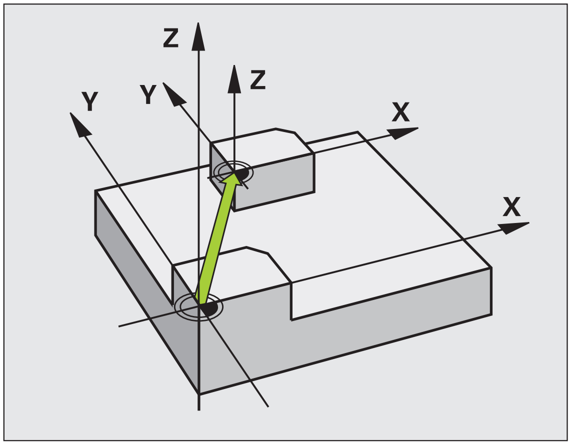
Effect

With the DATUM SETTING cycle you can activate as the new datum a preset defined in a preset table.

After a DATUM SETTING cycle definition, all of the coordinate inputs and datum shifts (absolute and incremental) are referenced to the new preset.

Status display

In the status display the TNC shows the active preset number behind the datum symbol.



Please note before programming:

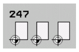
When activating a datum from the preset table, the TNC resets the active datum shift.

The TNC sets the preset only in the axes that are defined with values in the preset table. The datums of axes marked with – remain unchanged.

If you activate preset number 0 (line 0), then you activate the datum that you last set in a manual operating mode.

Cycle 247 is not functional in Test Run mode.

Cycle parameters



- **Number for datum?**: Enter the number of the datum to be activated from the preset table. Input range 0 to 65535

Example: NC blocks

```
13 CYCL DEF 247 DATUM SETTING
    Q339=4    ;DATUM NUMBER
```

11.5 MIRROR IMAGE (Cycle 8, DIN/ISO: G28)

Effect

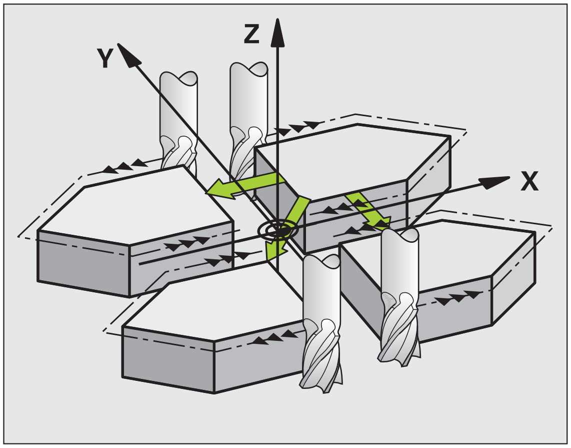
The TNC can machine the mirror image of a contour in the working plane.

The mirroring cycle becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active mirrored axes are shown in the additional status display.

- If you mirror only one axis, the machining direction of the tool is reversed (except in fixed cycles).
- If you mirror two axes, the machining direction remains the same.

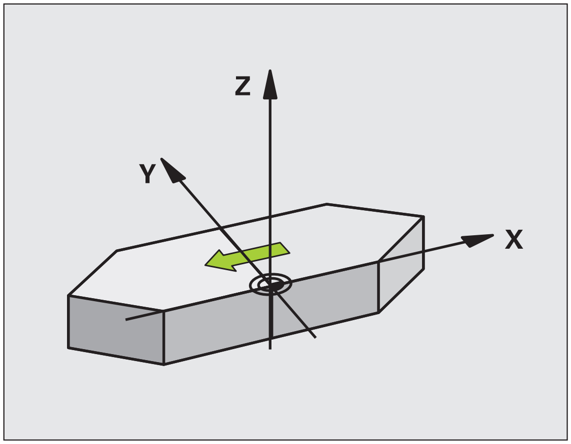
The result of the mirroring depends on the location of the datum:

- If the datum lies on the contour to be mirrored, the element simply flips over.
- If the datum lies outside the contour to be mirrored, the element also "jumps" to another location.



Reset

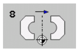
Program the MIRROR IMAGE cycle once again with NO ENT.



Please note while programming:

If you mirror only one axis, the machining direction is reversed for the milling cycles (Cycles 2xx). Exception: Cycle 208, in which the direction defined in the cycle applies.

Cycle parameters



- **Mirrored axis?**: Enter the axis to be mirrored. You can mirror all axes—including rotary axes—with the exception of the spindle axis and its associated auxiliary axis. You can enter up to three axes. Input range: Up to 3 NC axes **X**, **Y**, **Z**, **U**, **V**, **W**, **A**, **B**, **C**

Example: NC blocks

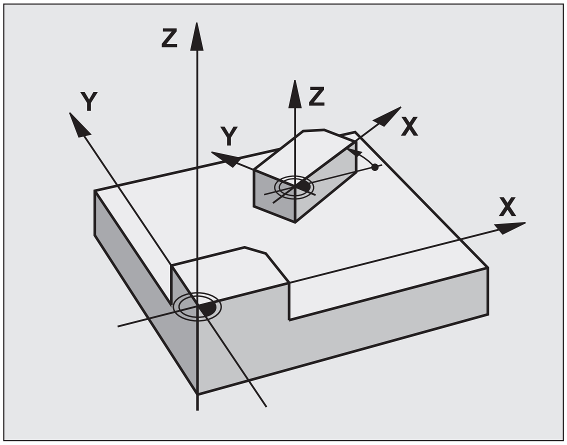
```
79 CYCL DEF 8.0 MIRROR IMAGE
80 CYCL DEF 8.1 X Y U
```

11.6 ROTATION (Cycle 10, DIN/ISO: G73)

Effect

The TNC can rotate the coordinate system about the active datum in the working plane within a program.

The ROTATION cycle becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active rotation angle is shown in the additional status display.

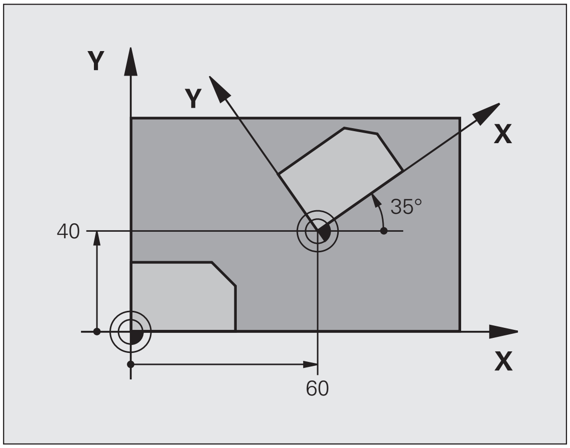


Reference axis for the rotation angle:

- X/Y plane: X axis
- Y/Z plane: Y axis
- Z/X plane: Z axis

Reset

Program the ROTATION cycle once again with a rotation angle of 0°.

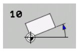


Please note while programming:

An active radius compensation is canceled by defining Cycle 10 and must therefore be reprogrammed, if necessary.

After defining Cycle 10, you must move both axes of the working plane to activate rotation for all axes.

Cycle parameters



- **Rotation**: Enter the rotation angle in degrees (°). Input range –360.000° to +360.000° (absolute or incremental)

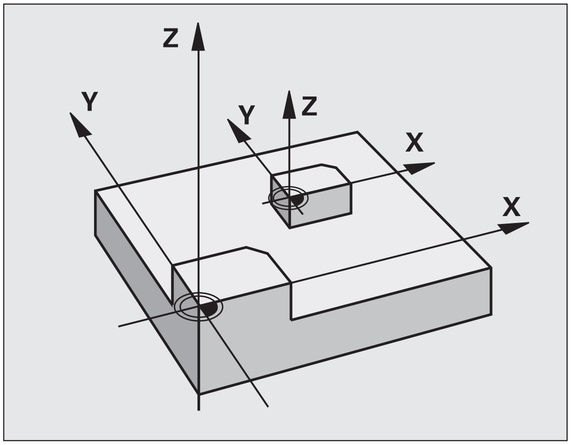
Example: NC blocks

```
12 CALL LBL 1
13 CYCL DEF 7.0 DATUM SHIFT
14 CYCL DEF 7.1 X+60
15 CYCL DEF 7.2 Y+40
16 CYCL DEF 10.0 ROTATION
17 CYCL DEF 10.1 ROT+35
18 CALL LBL 1
```

11.7 SCALING (Cycle 11, DIN/ISO: G72)

Effect

The TNC can increase or reduce the size of contours within a program, enabling you to program shrinkage and oversize allowances.



The SCALING FACTOR becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active scaling factor is shown in the additional status display.

The scaling factor has an effect on

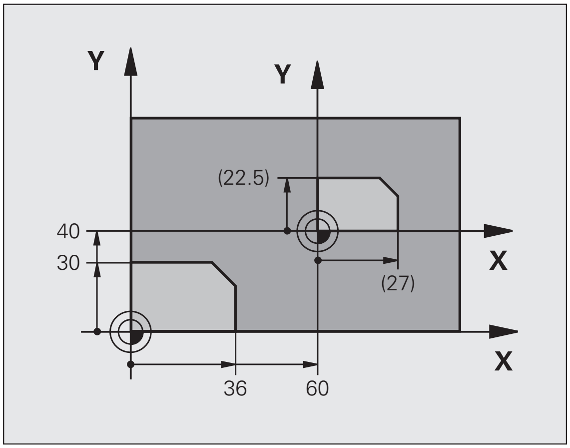
- the working plane, or on all three coordinate axes at the same time (depending on MP7410)
- dimensions in cycles
- the parallel axes U,V,W

Prerequisite

It is advisable to set the datum to an edge or a corner of the contour before enlarging or reducing the contour.

Enlargement: SCL greater than 1 (up to 99.999 999)

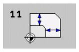
Reduction: SCL less than 1 (down to 0.000 001)



Reset

Program the SCALING cycle once again with a scaling factor of 1.

Cycle parameters



- **`Scaling factor?`**: Enter the scaling factor SCL. The TNC multiplies the coordinates and radii by the SCL factor (as described under "Effect" above). Input range: 0.000000 to 99.999999

Example: NC blocks

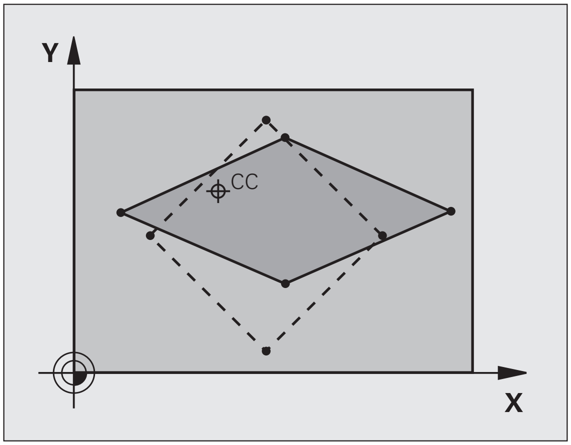
```
11 CALL LBL 1
12 CYCL DEF 7.0 DATUM SHIFT
13 CYCL DEF 7.1 X+60
14 CYCL DEF 7.2 Y+40
15 CYCL DEF 11.0 SCALING
16 CYCL DEF 11.1 SCL 0.75
17 CALL LBL 1
```

11.8 AXIS-SPECIFIC SCALING (Cycle 26)

Effect

With Cycle 26 you can account for shrinkage and oversize factors for each axis.

The SCALING FACTOR becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active scaling factor is shown in the additional status display.



Reset

Program the SCALING cycle once again with a scaling factor of 1 for the same axis.

Please note while programming:

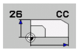
Coordinate axes sharing coordinates for arcs must be enlarged or reduced by the same factor.

You can program each coordinate axis with its own axis-specific scaling factor.

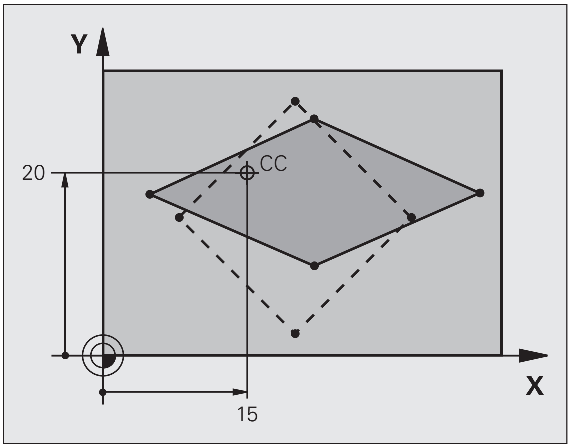
In addition, you can enter the coordinates of a center for all scaling factors.

The size of the contour is enlarged or reduced with reference to the center, and not necessarily (as in Cycle 11 SCALING) with reference to the active datum.

Cycle parameters



- **Axis and scaling factor**: Select the coordinate axis/axes by soft key and enter the factor(s) involved in enlarging or reducing. Input range: 0.000000 to 99.999999
- **Center coordinates**: Enter the center of the axis-specific enlargement or reduction. Input range -99999.9999 to 99999.9999



Example: NC blocks

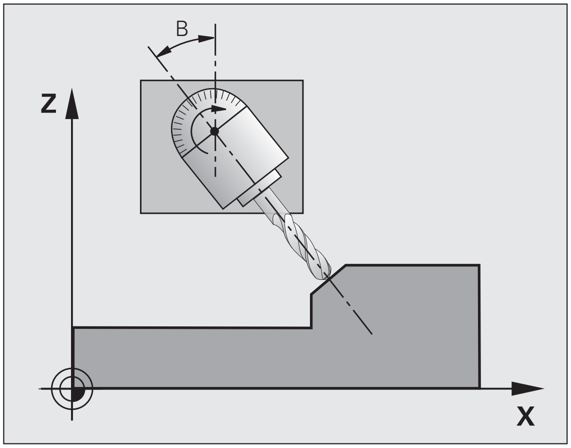
```
25 CALL LBL 1
26 CYCL DEF 26.0 AXIS-SPECIFIC SCALING
27 CYCL DEF 26.1 X 1.4 Y 0.6 CCX+15 CCY+20
28 CALL LBL 1
```

11.9 WORKING PLANE (Cycle 19, DIN/ISO: G80, Software Option 1)

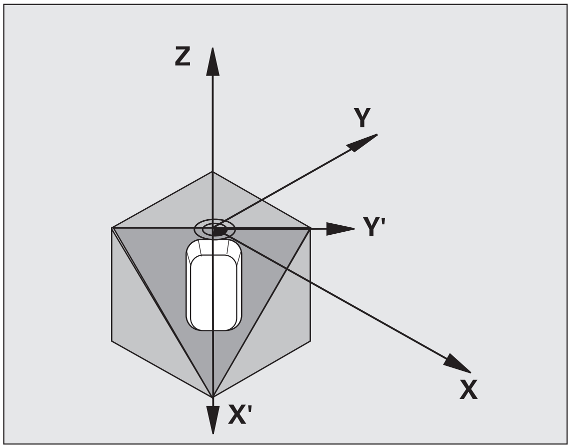
Effect

In Cycle 19 you define the position of the working plane—i.e. the position of the tool axis referenced to the machine coordinate system—by entering tilt angles. There are two ways to determine the position of the working plane:

- Enter the position of the rotary axes directly.
- Describe the position of the working plane using up to 3 rotations (spatial angle) of the **fixed machine** coordinate system. The required spatial angle can be calculated by cutting a perpendicular line through the tilted working plane and considering it from the axis around which you wish to tilt. With two spatial angles, every tool position in space can be defined exactly.



Note that the position of the tilted coordinate system, and therefore also all movements in the tilted system, are dependent on your description of the tilted plane.



If you program the position of the working plane via spatial angles, the TNC will calculate the required angle positions of the tilted axes automatically and will store these in the parameters Q120 (A axis) to Q122 (C axis).

Danger of collision!

Depending on your machine configuration, two mathematical solutions (axis positions) are possible for a spatial angle definition. Conduct appropriate tests on your machine to find out which axis position the TNC software selects in each case.

If the DCM software option is available to you, the axis position can be displayed in the PROGRAM + KINEMATICS view during test run (see User's Manual for Conversational Programming, **Dynamic collision monitoring**).

The axes are always rotated in the same sequence for calculating the tilt of the plane: The TNC first rotates the A axis, then the B axis, and finally the C axis.

Cycle 19 becomes effective as soon as it is defined in the program. As soon as you move an axis in the tilted system, the compensation for this specific axis is activated. You must move all axes to activate compensation for all axes.

If you set the function **`Tilting program run`** to **`Active`** in the Manual Operation mode, the angular value entered in this menu is overwritten by Cycle 19 WORKING PLANE.

Please note while programming:

The functions for tilting the working plane are interfaced to the TNC and the machine tool by the machine tool builder. For certain swivel heads and tilting tables the machine tool builder specifies whether the entered angles are interpreted as coordinates of the rotary axes or as mathematical angles of a tilted plane. Refer to your machine manual.

Because nonprogrammed rotary axis values are interpreted as unchanged, you should always define all three spatial angles, even if one or more angles are at zero.

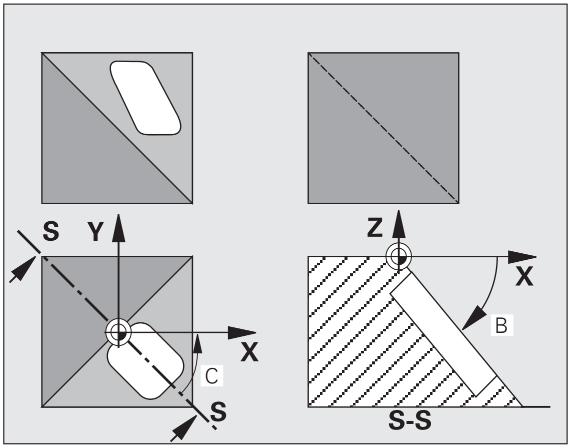
The working plane is always tilted around the active datum.

If you use Cycle 19 when M120 is active, the TNC automatically rescinds the radius compensation, which also rescinds the M120 function.

Danger of collision!

Ensure that the last defined angle is smaller than 360°.

Cycle parameters



- **Rotary axis and tilt angle?**: Enter the axes of rotation together with the associated tilt angles. The rotary axes A, B and C are programmed using soft keys. Input range -360.000 to 360.000

If the TNC automatically positions the rotary axes, you can enter the following parameters:

- **Feed rate? F=**: Traversing speed of the rotary axis during automatic positioning. Input range 0 to 99999.999
- **Set-up clearance?** (incremental): The TNC positions the swivel head so that the position that results from the extension of the tool by the set-up clearance does not change relative to the workpiece. Input range 0 to 99999.9999

Danger of collision!

Please note that the set-up clearance in Cycle 19 does not refer to the upper edge of the workpiece (as is the case in the fixed cycles) but rather to the active datum.

Reset

To cancel the tilt angles, redefine the WORKING PLANE cycle and enter an angular value of 0° for all axes of rotation. You must then program the WORKING PLANE cycle once again and respond to the dialog question with the NO ENT key to disable the function.

Positioning the axes of rotation

The machine tool builder determines whether Cycle 19 positions the axes of rotation automatically or whether they must be positioned manually in the program. Refer to your machine manual.

Manual positioning of rotary axes

If the rotary axes are not positioned automatically in Cycle 19, you must position them in a separate L block after the cycle definition.

If you use axis angles, you can define the axis values right in the L block. If you use spatial angles, then use the Q parameters **Q120** (A-axis value), **Q121** (B-axis value) and **Q122** (C-axis value), which are described by Cycle 19.

Example NC blocks:

`10 L Z+100 R0 FMAX`	
`11 L X+25 Y+10 R0 FMAX`	
`12 CYCL DEF 19.0 WORKING PLANE`	Define the spatial angle for calculation of the compensation
`13 CYCL DEF 19.1 A+0 B+45 C+0`	
`14 L A+Q120 C+Q122 R0 F1000`	Position the rotary axes by using values calculated by Cycle 19
`15 L Z+80 R0 FMAX`	Activate compensation for the spindle axis
`16 L X-8.5 Y-10 R0 FMAX`	Activate compensation for the working plane

For manual positioning, always use the rotary axis positions stored in Q parameters Q120 to Q122.

Avoid using functions, such as M94 (modulo rotary axes), in order to avoid discrepancies between the actual and nominal positions of rotary axes in multiple definitions.

Automatic positioning of rotary axes

If the rotary axes are positioned automatically in Cycle 19:

- The TNC can position only controlled axes
- In order for the tilted axes to be positioned, you must enter a feed rate and a set-up clearance in addition to the tilting angles, during cycle definition.
- Use only preset tools (the full tool length must be defined).
- The position of the tool tip as referenced to the workpiece surface remains nearly unchanged after tilting
- The TNC performs the tilt at the last programmed feed rate. The maximum feed rate that can be reached depends on the complexity of the swivel head or tilting table.

Example NC blocks:

`10 L Z+100 R0 FMAX`	
`11 L X+25 Y+10 R0 FMAX`	
`12 CYCL DEF 19.0 WORKING PLANE`	Define the angle for calculation of the compensation
`13 CYCL DEF 19.1 A+0 B+45 C+0 F5000 SETUP50`	Also define the feed rate and the clearance
`14 L Z+80 R0 FMAX`	Activate compensation for the spindle axis
`15 L X-8.5 Y-10 R0 FMAX`	Activate compensation for the working plane

Position display in the tilted system

On activation of Cycle 19, the displayed positions (**ACTL** and **NOML**) and the datum indicated in the additional status display are referenced to the tilted coordinate system. The positions displayed immediately after cycle definition might not be the same as the coordinates of the last programmed position before Cycle 19.

Workspace monitoring

The TNC monitors only those axes in the tilted coordinate system that are moved. If any of the software limit switches is traversed the TNC will display an error message.

Positioning in a tilted coordinate system

With the miscellaneous function M130 you can move the tool, while the coordinate system is tilted, to positions that are referenced to the non-tilted coordinate system.

Positioning movements with straight lines that are referenced to the machine coordinate system (blocks with M91 or M92) can also be executed in a tilted working plane. Constraints:

- Positioning is without length compensation.
- Positioning is without machine geometry compensation.
- Tool radius compensation is not permitted.

Combining coordinate transformation cycles

When combining coordinate transformation cycles, always make sure the working plane is swiveled around the active datum. You can program a datum shift before activating Cycle 19. In this case, you are shifting the machine-based coordinate system.

If you program a datum shift after having activated Cycle 19, you are shifting the tilted coordinate system.

Important: When resetting the cycles, use the reverse sequence used for defining them:

1. Activate the datum shift
2. Activate the tilting function
3. Activate rotation

...

Workpiece machining

...

1. Reset the rotation
2. Reset the tilting function
3. Reset the datum shift

Automatic workpiece measurement in the tilted system

The TNC measuring cycles enable you to have the TNC measure a workpiece in a tilted system automatically. The TNC stores the measured data in Q parameters for further processing (for example, for printout).

Procedure for working with Cycle 19 WORKING PLANE

1 Write the program

- Define the tool (not required if TOOL.T is active), and enter the full tool length.
- Call the tool
- Retract the tool in the tool axis to a position where there is no danger of collision with the workpiece or clamping devices during tilting.
- If required, position the rotary axis or axes with an L block to the appropriate angular value(s) (depending on a machine parameter).
- Activate datum shift if required.
- Define Cycle 19 WORKING PLANE; enter the angular values for the tilt axes
- Traverse all principal axes (X, Y, Z) to activate compensation.
- Write the program as if the machining process were to be executed in a non-tilted plane.
- If required, define Cycle 19 WORKING PLANE with other angular values to execute machining in a different axis position. In this case, it is not necessary to reset Cycle 19. You can define the new angular values directly.
- Reset Cycle 19 WORKING PLANE; program 0° for all tilt axes.
- Disable the WORKING PLANE function; redefine Cycle 19 and answer the dialog question with NO ENT.
- Reset datum shift if required.
- Position the tilt axes to the 0° position if required.

2 Clamp the workpiece

3 Preparations in the operating mode Positioning with Manual Data Input

Pre-position the rotary axis/axes to the corresponding angular value(s) for setting the datum. The angular value depends on the selected reference plane on the workpiece.

4 Preparations in the operating mode Manual Operation

Use the 3-D ROT soft key to set the function TILT WORKING PLANE to ACTIVE in the Manual Operating mode. For open loop axes, enter the angular values for the rotary axes into the menu.

If the axes are not controlled, the angular values entered in the menu must correspond to the actual position(s) of the rotary axis or axes, respectively. The TNC will otherwise calculate a wrong datum.

5 Datum setting

- Manually by touching the workpiece with the tool in the untilted coordinate system.
- Controlled with a HEIDENHAIN 3-D touch probe (see the Touch Probe Cycles User's Manual, chapter 2).
- Automatically with a HEIDENHAIN 3-D touch probe (see the Touch Probe Cycles User's Manual, chapter 3).

6 Start the part program in the operating mode Program Run, Full Sequence

7 Manual Operation mode

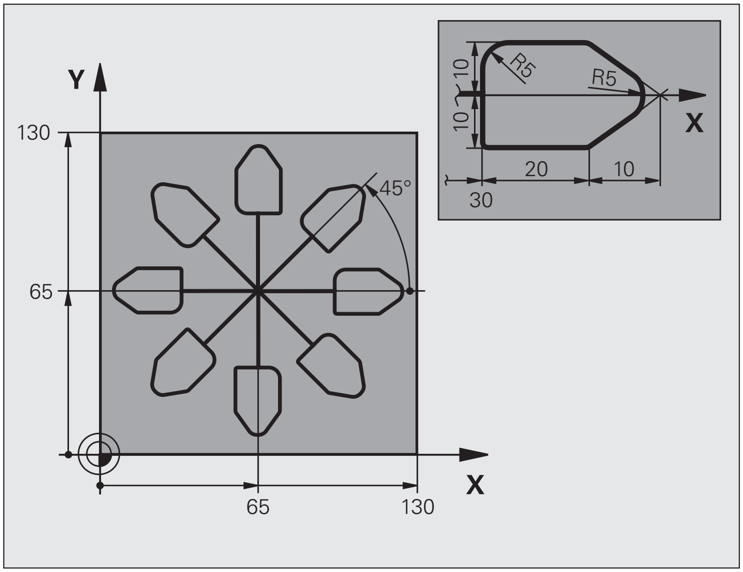
Use the 3-D ROT soft key to set the TILT WORKING PLANE function to INACTIVE. Enter an angular value of 0° for each rotary axis in the menu.

11.10Programming examples

Example: Coordinate transformation cycles

Program sequence

- Program the coordinate transformations in the main program
- Machining within a subprogram



`0 BEGIN PGM COTRANS MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-20`	Definition of workpiece blank
`2 BLK FORM 0.2 X+130 Y+130 Z+0`	
`3 TOOL DEF 1 L+0 R+1`	Tool definition
`4 TOOL CALL 1 Z S4500`	Tool call
`5 L Z+250 R0 FMAX`	Retract the tool
`6 CYCL DEF 7.0 DATUM SHIFT`	Shift datum to center
`7 CYCL DEF 7.1 X+65`	
`8 CYCL DEF 7.2 Y+65`	
`9 CALL LBL 1`	Call milling operation
`10 LBL 10`	Set label for program section repeat
`11 CYCL DEF 10.0 ROTATION`	Rotate by 45° (incremental)
`12 CYCL DEF 10.1 IROT+45`	
`13 CALL LBL 1`	Call milling operation
`14 CALL LBL 10 REP 6/6`	Return jump to LBL 10; repeat the milling operation six times
`15 CYCL DEF 10.0 ROTATION`	Reset the rotation
`16 CYCL DEF 10.1 ROT+0`	
`17 TRANS DATUM RESET`	Reset the datum shift

18 L Z+250 R0 FMAX M2	Retract the tool, end program
19 LBL 1	Subprogram 1
20 L X+0 Y+0 R0 FMAX	Define milling operation
21 L Z+2 R0 FMAX M3	
22 L Z-5 R0 F200	
23 L X+30 RL	
24 L IY+10	
25 RND R5	
26 L IX+20	
27 L IX+10 IY-10	
28 RND R5	
29 L IX-10 IY-10	
30 L IX-20	
31 L IY+10	
32 L X+0 Y+0 R0 F5000	
33 L Z+20 R0 FMAX	
34 LBL 0	
35 END PGM COTRANS MM	

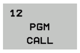
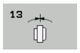
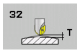
12

Cycles: Special Functions

12.1 Fundamentals

Overview

The TNC provides various cycles for the following special purposes:

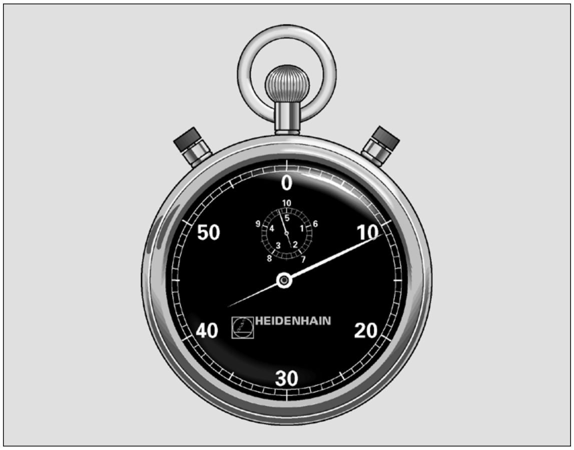
Cycle	Soft key	Page
9 DWELL TIME		Page 311
12 PROGRAM CALL		Page 312
13 SPINDLE ORIENTATION		Page 314
32 TOLERANCE		Page 315
225 ENGRAVING of texts		Page 319
290 INTERPOLATION TURNING (software option)		Page 323

12.2 DWELL TIME (Cycle 9, DIN/ISO: G04)

Function

This causes the execution of the next block within a running program to be delayed by the programmed DWELL TIME. A dwell time can be used for such purposes as chip breaking.

The cycle becomes effective as soon as it is defined in the program. Modal conditions such as spindle rotation are not affected.



Example: NC blocks

```
89 CYCL DEF 9.0 DWELL TIME
90 CYCL DEF 9.1 DWELL 1.5
```

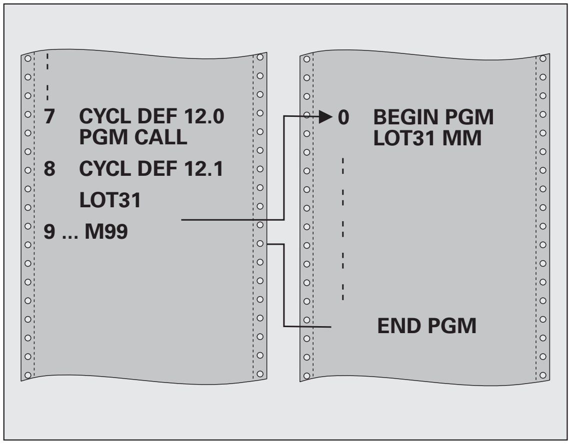
Cycle parameters

- **`Dwell time in seconds`**: Enter the dwell time in seconds. Input range: 0 to 3600 s (1 hour) in steps of 0.001 seconds

12.3 PROGRAM CALL (Cycle 12, DIN/ISO: G39)

Cycle function

Routines that you have programmed (such as special drilling cycles or geometrical modules) can be written as main programs. These can then be called like fixed cycles.



Please note while programming:

The program you are calling must be stored on the hard disk of your TNC.

If the program you are defining to be a cycle is located in the same directory as the program you are calling it from, you need only enter the program name.

If the program you are defining to be a cycle is not located in the same directory as the program you are calling it from, you must enter the complete path, for example **TNC:\KLAR35\FK1\50.H**.

If you want to define a DIN/ISO program to be a cycle, enter the file type .I behind the program name.

As a rule, Q parameters are globally effective when called with Cycle 12. So please note that changes to Q parameters in the called program can also influence the calling program.

Cycle parameters

12 PGM CALL

- **Program name**: Enter the name of the program you want to call and, if necessary, the directory it is located in. A maximum of 254 characters can be entered.

The following functions can be used to call the defined program:

- **CYCL CALL** (separate block) or
- **CYCL CALL POS** (separate block) or
- **M99** (blockwise) or
- **M89** (executed after every positioning block)

Example: Designate program 50 as a cycle and call it with M99

```
55 CYCL DEF 12.0 PGM CALL
56 CYCL DEF 12.1 PGM TNC:\KLAR35\FK1\50.H
57 L X+20 Y+50 FMAX M99
```

12.4 SPINDLE ORIENTATION (Cycle 13, DIN/ISO: G36)

Cycle function

Machine and TNC must be specially prepared by the machine tool builder for use of this cycle.

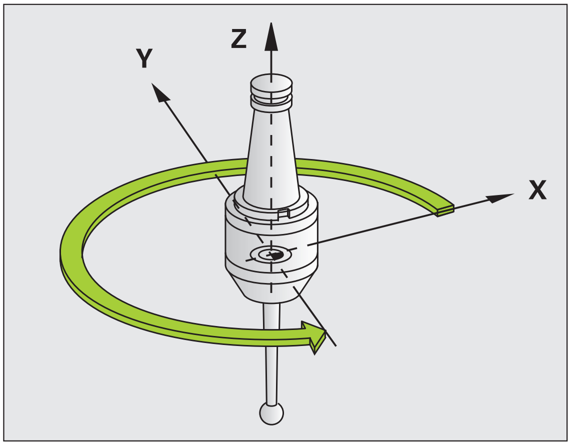
The TNC can control the machine tool spindle and rotate it to a given angular position.

Oriented spindle stops are required for

- Tool changing systems with a defined tool change position
- Orientation of the transmitter/receiver window of HEIDENHAIN 3-D touch probes with infrared transmission

The angle of orientation defined in the cycle is positioned to by entering M19 or M20 (depending on the machine).

If you program M19 or M20 without having defined Cycle 13, the TNC positions the machine tool spindle to an angle that has been set by the machine manufacturer (see your machine manual).



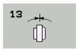
Example: NC blocks

```
93 CYCL DEF 13.0 ORIENTATION
94 CYCL DEF 13.1 ANGLE 180
```

Please note while programming:

Cycle 13 is used internally for Cycles 202, 204 and 209. Please note that, if required, you must program Cycle 13 again in your NC program after one of the machining cycles mentioned above.

Cycle parameters

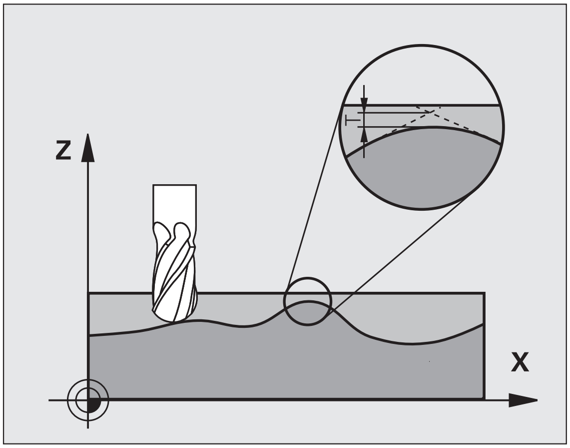


- **Angle of orientation**: Enter the angle referenced to the reference axis of the working plane. Input range: 0.0000° to 360.0000°

12.5 TOLERANCE (Cycle 32, DIN/ISO: G62)

Cycle function

> Machine and TNC must be specially prepared by the machine tool builder for use of this cycle. The cycle may be locked.



With the entries in Cycle 32 you can influence the result of HSC machining with respect to accuracy, surface definition and speed, inasmuch as the TNC has been adapted to the machine's characteristics.

The TNC automatically smoothes the contour between two path elements (whether compensated or not). The tool has constant contact with the workpiece surface and therefore reduces wear on the machine tool. The tolerance defined in the cycle also affects the traverse paths on circular arcs.

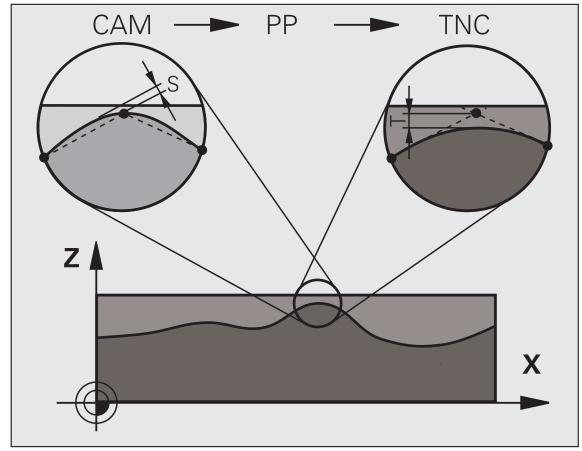
If necessary, the TNC automatically reduces the programmed feed rate so that the program can be machined at the fastest possible speed without short pauses for computing time. **Even if the TNC does not move with reduced speed, it will always comply with the tolerance that you have defined.** The larger you define the tolerance, the faster the TNC can move the axes.

Smoothing the contour results in a certain amount of deviation from the contour. The size of this contour error (**`tolerance value`**) is set in a machine parameter by the machine manufacturer. You can change the pre-set tolerance value with Cycle **32**.

Influences of the geometry definition in the CAM system

The most important factor of influence in offline NC program creation is the chord error S defined in the CAM system. The maximum point spacing of NC programs generated in a post processor (PP) is defined through the chord error. If the chord error is less than or equal to the tolerance value **T** defined in Cycle 32, then the TNC can smooth the contour points unless any special machine settings limit the programmed feed rate.

You will achieve optimal smoothing if in Cycle 32 you choose a tolerance value **T** that is at least twice as large as chord error specified in the CAM system.



Please note while programming:

With very small tolerance values the machine cannot cut the contour without jerking. These jerking movements are not caused by poor processing power in the TNC, but by the fact that, in order to machine the contour element transitions very exactly, the TNC might have to drastically reduce the speed.

Cycle 32 is DEF active which means that it becomes effective as soon as it is defined in the part program.

The TNC resets Cycle 32 if you

- Redefine it and confirm the dialog question for the **`tolerance value`** with NO ENT.
- Select a new program with the PGM MGT key.

After you have reset Cycle 32, the TNC reactivates the tolerance that was predefined by machine parameter.

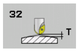
In a program with millimeters set as unit of measure, the TNC interprets the entered tolerance value in millimeters. In an inch program it interprets it as inches.

If you load a program with Cycle 32 that contains only the cycle parameter **`Tolerance value`** T, the TNC inserts the two remaining parameters with the value 0 if required.

As the tolerance value increases, the diameter of circular movements usually decreases. If the HSC filter is active on your machine (ask your machine tool builder, if necessary), the circle can also become larger.

If Cycle 32 is active, the TNC shows the parameters defined for Cycle 32 on the **`CYC`** tab of the additional status display.

Cycle parameters



- **`Tolerance value T`**: Permissible contour deviation in mm (or inches with inch programming). Input range 0 to 99999.9999
- **`HSC MODE, Finishing=0, Roughing=1`**: Activate filter:
  - Input value 0:
    Milling with increased contour accuracy. The TNC uses internally defined finishing filter settings
  - Input value 1:
    Milling at an increased feed rate. The TNC uses internally defined roughing filter settings
- **`Tolerance for rotary axes TA`**: Permissible position error of rotary axes in degrees when **`M128`** is active (**`TCPM FUNCTION`**). The TNC always reduces the feed rate in such a way that—if more than one axis is traversed—the slowest axis moves at its maximum feed rate. Rotary axes are usually much slower than linear axes. You can significantly reduce the machining time for programs for more than one axis by entering a large tolerance value (e.g. 10°), since the TNC does not always have to move the rotary axis to the given nominal position. The contour will not be damaged by entering a rotary axis tolerance value. Only the position of the rotary axis with respect to the workpiece surface will change. Input range 0 to 179.9999

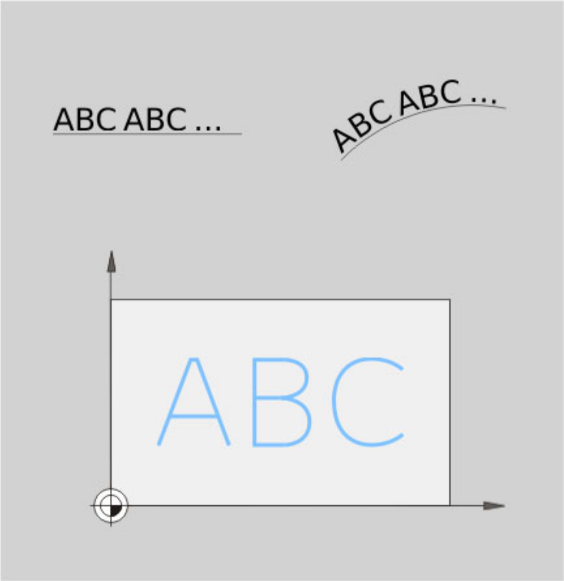
Example: NC blocks

```
95 CYCL DEF 32.0 TOLERANCE
96 CYCL DEF 32.1 T0.05
97 CYCL DEF 32.2 HSC-MODE:1 TA5
```

12.6 ENGRAVING (Cycle 225, DIN/ISO: G225)

Cycle run

This cycle is used to engrave texts on a flat surface of the workpiece. The texts can be arranged in a straight line or along an arc.



1. The TNC positions the tool in the working plane to the starting point of the first character.
2. The tool plunges perpendicularly to the engraving floor and mills the character. The TNC retracts the tool to the set-up clearance between the characters when required. At the end of the character the tool is at the set-up clearance above the workpiece surface.
3. This process is repeated for all characters to be engraved.
4. Finally, the TNC retracts the tool to the 2nd set-up clearance.

Please note while programming:

The algebraic sign for the cycle parameter DEPTH determines the working direction.

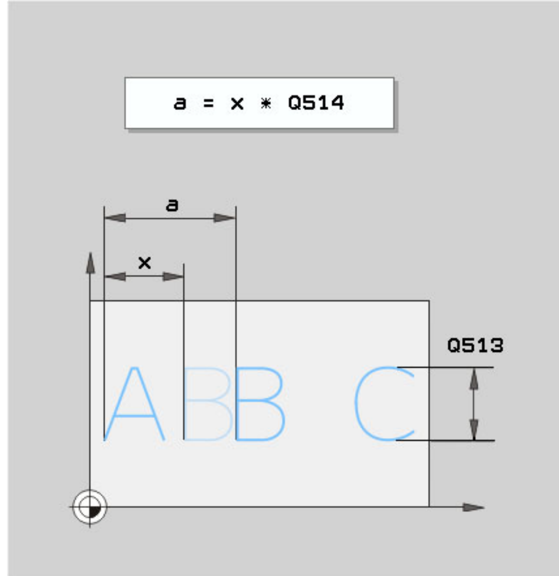
If you engrave the text in a straight line (**Q516=0**), the starting point of the first character is determined by the tool position at the time the cycle is called.

If you engrave the text along an arc (**Q516=1**), the arc's center is determined by the tool position at the time the cycle is called.

The text to be engraved can also be transferred with a string variable (**QS**).

Cycle parameters

- **Engraving text** QS500: Text to be engraved inside quotation marks. Assignment of a string variable through the Q key of the numerical keypad. The Q key on the ASCI keyboard represents normal text input. Maximum 256 characters permitted; permissible characters: See "Engraving system variables" on page 322
- **Character height** Q513 (absolute): Height of the characters to be engraved in mm. Input range 0 to 99999.9999
- **Space factor** Q514: The font used is a proportional font. Each character has its own width, which is engraved correspondingly by the TNC if you program Q514 = 0. If Q514 is not equal to 0, the TNC scales the space between the characters. Input range 0 to 9.9999
- **Font** Q515: Currently without function
- **Text on a line/on an arc (0/1)** Q516:
  Engrave the text in a straight line: Input = 0
  Engrave the text on an arc: Input = 1
- **Angle of rotation** Q374: Center angle if the text is to be arranged on an arc. Engraving angle when text is in a straight line. Input range -360.0000 to +360.0000°
- **Radius of text on an arc** Q517 (absolute): Radius of the arc in mm on which the TNC is to arrange the text. Input range 0 to 99999.9999
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while engraving. Input range 0 to 99999.999; alternatively **FAUTO**, **FU** or **FZ**
- **Depth** Q201 (incremental value): Distance between workpiece surface and engraving floor
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min when moving into the workpiece. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999; alternatively **PREDEF**



Example: NC blocks

```
62 CYCL DEF 225 ENGRAVING
    QS500="TXT2";ENGRAVING TEXT
    Q513=10   ;CHARACTER HEIGHT
    Q514=0    ;SPACE FACTOR
    Q515=0    ;FONT
    Q516=0    ;TEXT LAYOUT
    Q374=0    ;ANGLE OF ROTATION
    Q515=0    ;CIRCLE RADIUS
    Q207=750  ;FEED RATE FOR MILLING
    Q201=-0.5 ;DEPTH
    Q206=150  ;FEED RATE FOR PLNGNG
    Q200=2    ;SET-UP CLEARANCE
    Q203=+20  ;SURFACE COORDINATE
    Q204=50   ;2ND SET-UP CLEARANCE
```

Allowed engraving characters

The following special characters are allowed in addition to lowercase letters, uppercase letters and numbers:

)! # $ % & ' ( ) * + , - . / : ; < = > ? @ [\ ] _

> The TNC uses the special characters `%` and `\` for special functions. These characters must be indicated twice in the text to be engraved (e.g. `%%`) if you want to engrave them.

You can also use the cycle to engrave German umlauts and the diameter symbol::

Character	Input
m	%ae
ö	%oe
ü	%ue
Ä	%AE
Ö	%OE
Ü	%UE
ø	%D

Characters that cannot be printed

Apart from text, you can also define certain non-printable characters for formatting purposes. Enter the special character **** before the non-printable characters.

The following formatting possibilities are available:

- **\n**: Line break
- **\t**: Horizontal tab (the tab width is permanently set to 8 characters)
- **\v**: Vertical tab (the tab width is permanently set to one line)

Engraving system variables

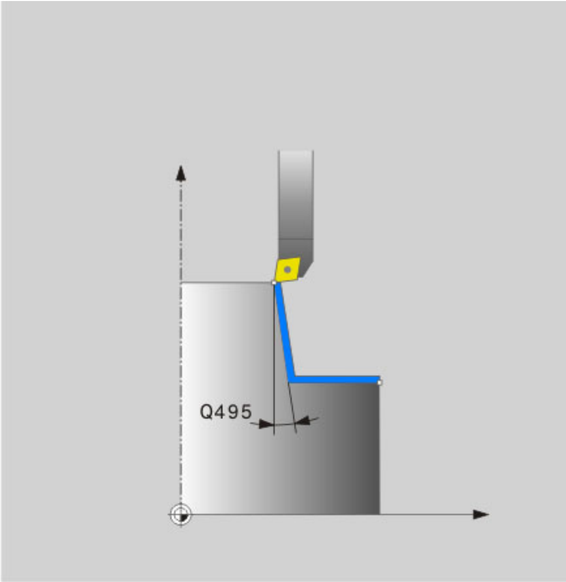
In addition to the standard characters, you can engrave the contents of certain system variables. Enter the special character **%** before the system variable.

You can also engrave the current date. Enter **%time<x>**. **<x>** defines the date format whose meaning is identical to the function **SYSSTR ID332** (see the User's Manual for Conversational Programming, "Q parameter programming" chapter, "Copying system data to a string" section).

Keep in mind that you must enter a leading 0 when entering the date formats 1 to 9, e.g. **time08**.

12.7 INTERPOLATION TURNING (software option, Cycle 290, DIN/ISO: G290)

Cycle run



This cycle is used to create a rotationally symmetric shoulder or a recess in the working plane, which are defined by the starting and end point (see also "Machining variants" on page 327). The center of rotation is the starting point (XY) at the time the cycle is called. The rotational surfaces can be inclined or rounded relative to each other. Interpolation-turning or milling cycles can be used to machine the surfaces.

The workpiece does not rotate during interpolation turning. The tool moves in a circular motion in the principal axes X and Y. At the same time, the TNC orients the spindle S so that the cutting edge of the turning tool is always oriented to the center of rotation of the workpiece. Cycle 290 can thus also be used on a three-axis machine.

The center of the machining operation does not need to be in the center of a rotary table. The center of the machining operation is defined by the tool position at the time the cycle is called.

1 The TNC moves the tool at clearance height to the starting point of machining. The starting point is obtained by extending the contour starting point tangentially by the set-up clearance.

2 The TNC uses the interpolation turning cycle to machine the defined contour. In interpolation turning the principal axes of the working plane move on a circle, whereas the spindle axis is oriented perpendicularly to the surface.

6 At the end point of the contour, the TNC retracts the tool perpendicularly by the set-up clearance.

4 Finally, the TNC retracts the tool to the clearance height.

Please note while programming:

You can use a turning tool or a milling tool (Q444=0) for this cycle. The geometry data of this tool are defined in the TOOL.T tool table as follows:

- Column **L** (**DL** for compensation values):
  Length of the tool (bottommost point of the tool cutting edge)
- Column **R** (**DR** for compensation values):
  Effective radius of the tool (outermost point of the tool cutting edge)
- Column **R2** (**DR2** for compensation values):
  Cutting-edge radius of the tool

Machine and TNC must be specially prepared by the machine tool builder for use of this cycle. Refer to your machine manual.

This cycle is effective only for machines with servo-controlled spindle (exception: **Q444=0**).

Software option 96 must be enabled.

Roughing operations with multiple passes are not possible in this cycle.

The center of interpolation is the tool position at the time the cycle is called.

The TNC extends the first surface to be machined by the set-up clearance.

You can use the values **DL** and **DR** of the **TOOL CALL** block to realize oversizes. **DR2** entries in the **TOOL CALL** block are not taken into account by the TNC.

Before cycle call, define a large tolerance with Cycle 32 for your machine to attain high contour speeds.

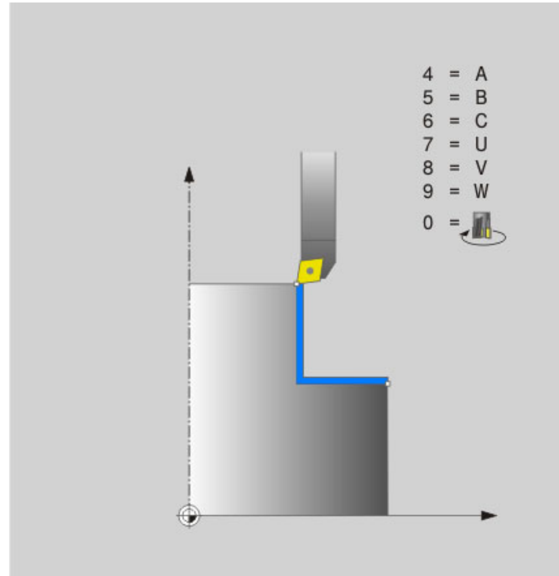
Program a cutting speed that can just be reached at the contour speed of the machine axes. This ensures optimum geometry resolution and a constant machining speed.

The TNC does not monitor for possible damage to the contour, which might be caused by unsuitable tool geometry.

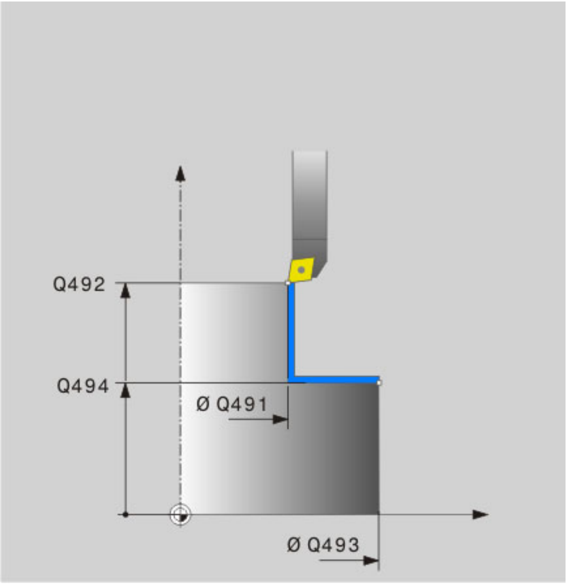
Note the machining variants: See "Machining variants" on page 327

Cycle parameters

- **Set-up clearance** Q200 (incremental value): Extension of the defined contour during approach and departure. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Clearance height** Q445 (absolute): Absolute height at which the tool cannot collide with the workpiece. Position for tool retraction at the end of the cycle. Input range -99999.9999 to 99999.9999
- **Angle for spindle orientation** Q336 (absolute): Angle for orienting the cutting edge to the 0° position of the spindle. Input range -360.0000 to 360.0000
- **Cutting speed [m/min]** Q440: Cutting speed of the tool in m/min. Input range 0 to 99.999
- **Infeed per revolution [mm/rev]** Q441: Feed rate of the tool per revolution. Input range 0 to 99.999
- **Start angle in plane XY** Q442: Starting angle in the XY plane. Input range 0 to 359.999
- **Machining direction (–1/+1)** Q443:
  Machine in clockwise direction: Input = –1
  Machine in counterclockwise direction: Input = +1
- **Interpolating axis (4...9)** Q444: Axis designation of the interpolating axis.
  A axis is interpolating axis: Input = 4
  B axis is interpolating axis: Input = 5
  C axis is interpolating axis: Input = 6
  U axis is interpolating axis: Input = 7
  V axis is interpolating axis: Input = 8
  W axis is interpolating axis: Input = 9
  Contour milling: Input = 0



- **`Diameter at contour start`** Q491 (absolute): Corner of starting point in X, enter the diameter. Input range -99999.9999 to 99999.9999
- **`Contour start in Z`** Q492 (absolute): Corner of the starting point in Z. Input range 99999.9999 to 99999.9999
- **`Diameter at end of contour`** Q493 (absolute): Corner of end point in X, enter the diameter. Input range -99999.9999 to 99999.9999
- **`Contour end in Z`** Q494 (absolute): Corner of the end point in Z. Input range 99999.9999 to 99999.9999
- **`Angle of circumferential surface`** Q495: Angle of the first surface to be machined in degrees. Input range -179.999 to 179.999
- **`Angle of the face`** Q496: Angle of the second surface to be machined in degrees. Input range -179.999 to 179.999
- **`Radius of contour edge`** Q500: Corner rounding between the surfaces to be machined. Input range 0 to 999.999



Example: NC blocks

```
62 CYCL DEF 290 INTERPOLATION TURNING
     Q200=2     ;SET-UP CLEARANCE
     Q445=+50   ;CLEARANCE HEIGHT
     Q336=0     ;ANGLE OF SPINDLE
     Q440=20    ;CUTTING SPEED
     Q441=0.75  ;INFEED
     Q442=+0    ;STARTING ANGLE
     Q443=-1    ;MACHINING DIRECTION
     Q444=+6    ;INTERPOLATED AXIS
     Q491=+25   ;DIAMETER AT CONTOUR START
     Q492=+0    ;CONTOUR START IN Z
     Q493=+50   ;CONTOUR END IN X
     Q494=-45   ;CONTOUR END IN Z
     Q495=+0    ;ANGLE OF CYLINDER SURFACE
     Q496=+0    ;ANGLE OF FACE
     Q500=4.5   ;RADIUS OF CONTOUR EDGE
```

Contour milling

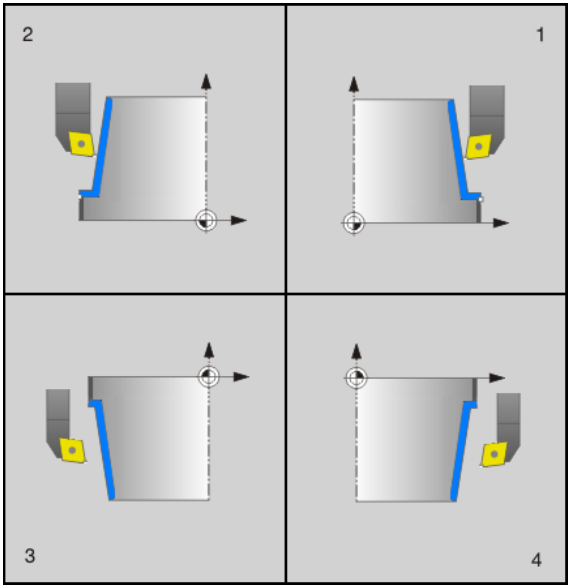
You can mill the surfaces by entering **Q444=0**. Use a milling cutter with a cutting radius (R2) for this machining operation. It is usually advisable to pre-machine surfaces with a large oversize by milling rather than by interpolation turning.

> Milling operations with multiple passes are possible in this cycle.
>
> Keep in mind that the feed rate during milling matches the value specified in **Q440** (cutting speed). The cutting speed is specified in meters per minute.

Machining variants

Combining the starting and end points with the angles Q495 and Q496 results in the following possible machining operations:

- **Outside machining in quadrant 1 (1)**:
  - Enter the circumferential angle (Q495) as a positive value.
  - Enter the angle of the face (Q496) as a negative value.
  - For the contour start in X (Q491), enter a value smaller than the contour end in X (Q493).
  - For the contour start in Z (Q492), enter a value greater than the contour end in Z (Q494).
- **Inside machining in quadrant 2 (2)**:
  - Enter the circumferential angle (Q495) as a negative value.
  - Enter the angle of the face (Q496) as a positive value.
  - For the contour start in X (Q491), enter a value greater than the contour end in X (Q493).
  - For the contour start in Z (Q492), enter a value greater than the contour end in Z (Q494).



- **Outside machining in quadrant 3 (3)**:
  - Enter the circumferential angle (Q495) as a positive value.
  - Enter the angle of the face (Q496) as a negative value.
  - For the contour start in X (Q491), enter a value greater than the contour end in X (Q493).
  - For the contour start in Z (Q492), enter a value smaller than the contour end in Z (Q494).
- **Inside machining in quadrant 4 (4)**:
  - Enter the circumferential angle (Q495) as a negative value.
  - Enter the angle of the face (Q496) as a positive value.
  - For the contour start in X (Q491), enter a value smaller than the contour end in X (Q493).
  - For the contour start in Z (Q492), enter a value smaller than the contour end in Z (Q494).
- **Recess axial**:
  - For the contour start in X (Q491), enter a value equal to the contour end in X (Q493).
- **Recess radial**:
  - For the contour start in Z (Q492), enter a value smaller than the contour end in Z (Q494).

13

Using Touch Probe Cycles

13.1 General information about touch probe cycles

The TNC must be specially prepared by the machine tool builder for the use of a 3-D touch probe. The machine manual provides further information.

Please note that HEIDENHAIN grants a warranty for the function of the touch probe cycles only if HEIDENHAIN touch probes are used!

If you are carrying out measurements during program run, be sure that the tool data (length, radius) can be used from the calibrated data or from the last **TOOL CALL** block (selected with MP7411).

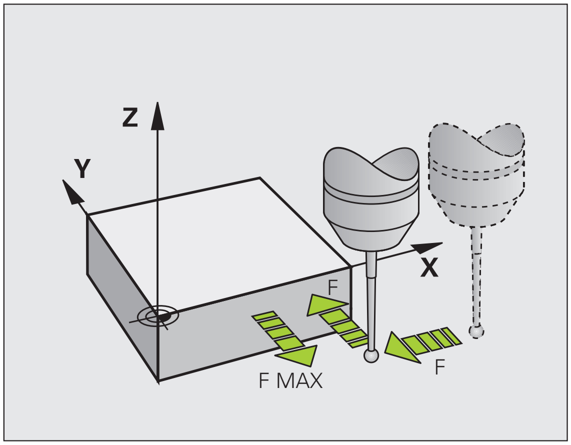
Principle of function

Whenever the TNC runs a touch probe cycle, the 3-D touch probe approaches the workpiece in one linear axis. This is also true during an active basic rotation or with a tilted working plane. The machine tool builder determines the probing feed rate in a machine parameter (see "Before You Start Working with Touch Probe Cycles" later in this chapter).

When the probe stylus contacts the workpiece,

- the 3-D touch probe transmits a signal to the TNC: the coordinates of the probed position are stored,
- the touch probe stops moving, and
- returns to its starting position at rapid traverse.

If the stylus is not deflected within a distance defined in MP6130, the TNC displays an error message.



Touch probe cycles in the Manual Operation and Electronic Handwheel modes

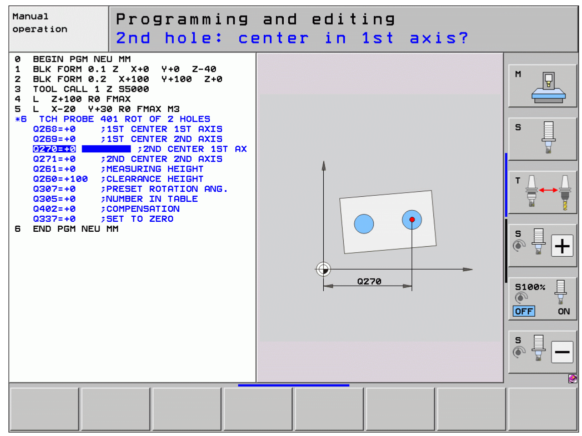
In the Manual Operation and El. Handwheel modes, the TNC provides touch probe cycles that allow you to:

- Calibrate the touch probe
- Compensate workpiece misalignment
- Set datums

Touch probe cycles for automatic operation

Besides the touch probe cycles, which you can use in the Manual and El. Handwheel modes, the TNC provides numerous cycles for a wide variety of applications in automatic mode:

- Calibrating a touch trigger probe
- Compensating workpiece misalignment
- Setting datums
- Automatic workpiece inspection
- Automatic tool measurement



You can program the touch probe cycles in the Programming and Editing operating mode via the TOUCH PROBE key. Like the most recent fixed cycles, touch probe cycles with numbers greater than 400 use Q parameters as transfer parameters. Parameters with specific functions that are required in several cycles always have the same number: For example, Q260 is always assigned the clearance height, Q261 the measuring height, etc.

To simplify programming, the TNC shows a graphic during cycle definition. In the graphic, the parameter that needs to be entered is highlighted (see figure at right).

Defining the touch probe cycle in the Programming and Editing mode of operation

TOUCH PROBE

- The soft-key row shows all available touch probe functions divided into groups.
- Select the desired probe cycle group, for example datum setting. Cycles for automatic tool measurement are available only if your machine has been prepared for them.
- Select a cycle, e.g. datum setting at pocket center. The TNC initiates the programming dialog and asks for all required input values. At the same time a graphic of the input parameters is displayed in the right screen window. The parameter that is asked for in the dialog prompt is highlighted.
- Enter all parameters requested by the TNC and conclude each entry with the ENT key.
- The TNC ends the dialog when all required data has been entered

Group of measuring cycles	Soft key	Page
Cycles for automatic measurement and compensation of workpiece misalignment		Page 338
Cycles for automatic workpiece presetting		Page 360
Cycles for automatic workpiece inspection		Page 414
Calibration cycles, special cycles	SPECIAL CYCLES	Page 464
Cycles for automatic kinematics measurement	KINEMATICS	Page 480
Cycles for automatic tool measurement (enabled by the machine tool builder)		Page 512

Example: NC blocks

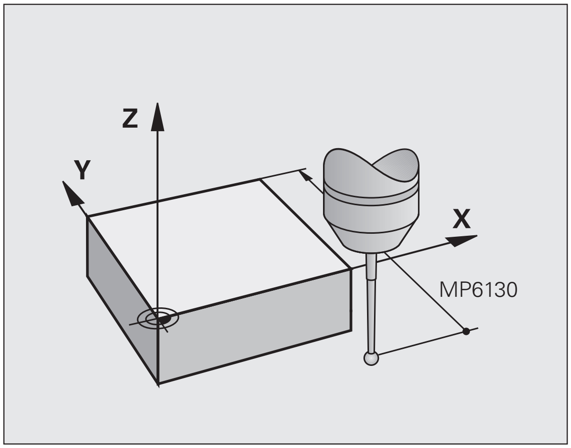
```
5 TCH PROBE 410 DATUM INSIDE RECTAN.
    Q321=+50  ;CENTER IN 1ST AXIS
    Q322=+50  ;CENTER IN 2ND AXIS
    Q323=60   ;1ST SIDE LENGTH
    Q324=20   ;2ND SIDE LENGTH
    Q261=-5   ;MEASURING HEIGHT
    Q320=0    ;SET-UP CLEARANCE
    Q260=+20  ;CLEARANCE HEIGHT
    Q301=0    ;MOVE TO CLEARANCE
    Q305=10   ;NO. IN TABLE
    Q331=+0   ;DATUM
    Q332=+0   ;DATUM
    Q303=+1   ;MEAS. VALUE TRANSFER
    Q381=1    ;PROBE IN TS AXIS
    Q382=+85  ;1ST CO. FOR TS AXIS
    Q383=+50  ;2ND CO. FOR TS AXIS
    Q384=+0   ;3RD CO. FOR TS AXIS
    Q333=+0   ;DATUM
```

13.2 Before you start working with touch probe cycles

To make it possible to cover the widest possible range of applications, machine parameters enable you to determine the behavior common to all touch probe cycles.

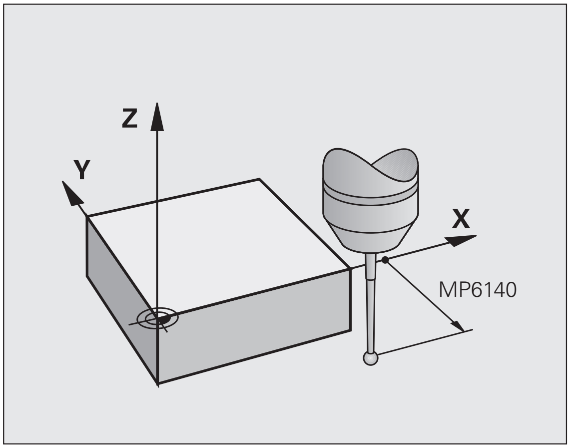
Maximum traverse to touch point: MP6130

If the stylus is not deflected within the path defined in MP6130, the TNC outputs an error message.



Safety clearance to touch point: MP6140

In MP6140 you define how far from the defined (or calculated) touch point the TNC is to pre-position the touch probe. The smaller the value you enter, the more exactly you must define the touch point position. In many touch probe cycles you can also define a safety clearance that is added to MP6140.



Orient the infrared touch probe to the programmed probe direction: MP6165

To increase measuring accuracy, you can use MP 6165 = 1 to have an infrared touch probe oriented in the programmed probe direction before every probe process. In this way the stylus is always deflected in the same direction.

> If you change MP6165, you must recalibrate the touch probe, because its deflection behavior changes.

Consider a basic rotation in the Manual Operation mode: MP6166

Set MP 6166 = 1 for the TNC to consider an active basic rotation during the probing process (the workpiece is approached along an angular path if required) to ensure that the measuring accuracy for probing individual positions is also increased in Setup mode.

This feature is not active during the following functions in the Manual Operation mode:

- Calibrate length
- Calibrate radius
- Measure basic rotation

Multiple measurements: MP6170

To increase measuring certainty, the TNC can run each probing process up to three times in sequence. If the measured position values differ too greatly, the TNC outputs an error message (the limit value is defined in MP6171). With multiple measurements it is possible to detect random errors, e.g. from contamination.

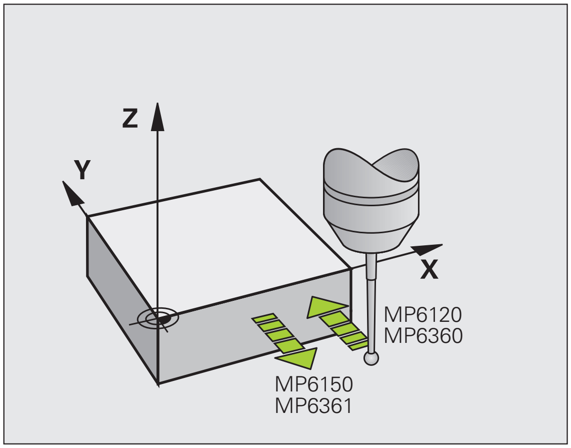
If the measured values lie within the confidence interval, the TNC saves the mean value of the measured positions.

Confidence interval for multiple measurements: MP6171

In MP6171 you store the value by which the results may differ when you make multiple measurements. If the difference in the measured values exceeds the value in MP6171, the TNC outputs an error message.

Touch trigger probe, probing feed rate: MP6120

In MP6120 you define the feed rate at which the TNC is to probe the workpiece.



Touch trigger probe, rapid traverse for positioning: MP6150

In MP6150 you define the feed rate at which the TNC pre-positions the touch probe, or positions it between measuring points.

Touch trigger probe, rapid traverse for positioning: MP6151

In MP6151 you define whether the TNC is to position the touch probe at the feed rate defined in MP6150 or at rapid traverse.

- Input value = 0: Position at feed rate from MP6150
- Input value = 1: Pre-position at rapid traverse

KinematicsOpt: Tolerance limit in Optimization mode: MP6600

In **MP6600** you define the tolerance limit starting from which the TNC displays a note in the Optimization mode when the measured kinematic data is greater than this limit value. The default value is 0.05. The larger the machine, the greater these values should be.

- Input range 0.001 to 0.999

KinematicsOpt, permissible deviation of the calibration ball radius: MP6601

In **MP6601** you define the maximum permissible deviation from the entered cycle parameter by the calibration ball radius measured in the cycles.

- Input range: 0.01 to 0.1

The TNC calculates the calibration ball radius twice at every measuring point for all 5 touch points. If the radius is greater than Q407 + MP6601 an error message appears because it could be contamination.

If the radius found by the TNC is less than 5 * (Q407 - MP6601), the TNC also issues an error message.

Executing touch probe cycles

All touch probe cycles are DEF active. This means that the TNC runs the cycle automatically as soon as the TNC executes the cycle definition in the program run.

> Make sure that at the beginning of the cycle the compensation data (length, radius) from the calibrated data or from the last TOOL CALL block are active (selection via MP7411, see the User's Manual of the iTNC530, "General User Parameters").
>
> You can also run the Touch Probe Cycles 408 to 419 during an active basic rotation. Make sure, however, that the basic rotation angle does not change when you use Cycle 7 DATUM SHIFT with datum tables after the measuring cycle.

Touch probe cycles with a number greater than 400 position the touch probe according to a positioning logic:

- If the current coordinate of the south pole of the stylus is less than the coordinate of the clearance height (defined in the cycle), the TNC retracts the touch probe in the touch-probe axis to the clearance height and then positions it in the working plane to the first probe point.
- If the current coordinate of the stylus south pole is greater than the coordinate of the clearance height, then the TNC first positions the touch probe to the first probe point in the working plane, and then in the touch-probe axis directly to the measuring height.

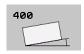
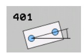
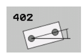
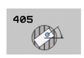
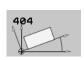
14

Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment

14.1 Fundamentals

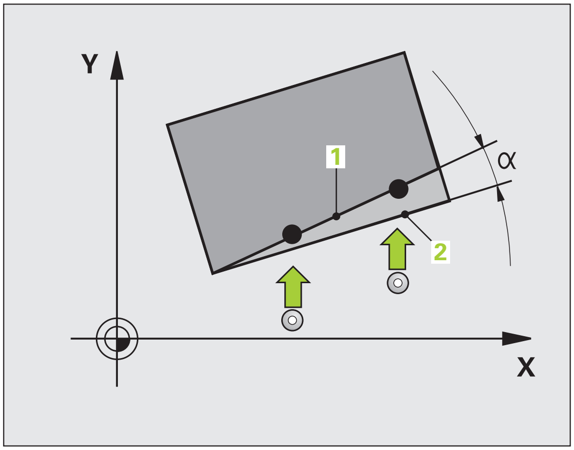
Overview

The TNC provides five cycles that enable you to measure and compensate workpiece misalignment. In addition, you can reset a basic rotation with Cycle 404:

Cycle	Soft key	Page
400 BASIC ROTATION Automatic measurement using two points. Compensation via basic rotation.		Page 340
401 ROT OF 2 HOLES Automatic measurement using two holes. Compensation via basic rotation.		Page 343
402 ROT OF 2 STUDS Automatic measurement using two studs. Compensation via basic rotation.		Page 346
403 ROT IN ROTARY AXIS Automatic measurement using two points. Compensation via table rotation.		Page 349
405 ROT IN C AXIS Automatic alignment of an angular offset between a hole center and the positive Y axis. Compensation via table rotation.		Page 354
404 SET BASIC ROTATION Setting any basic rotation.		Page 353

Characteristics common to all touch probe cycles for measuring workpiece misalignment

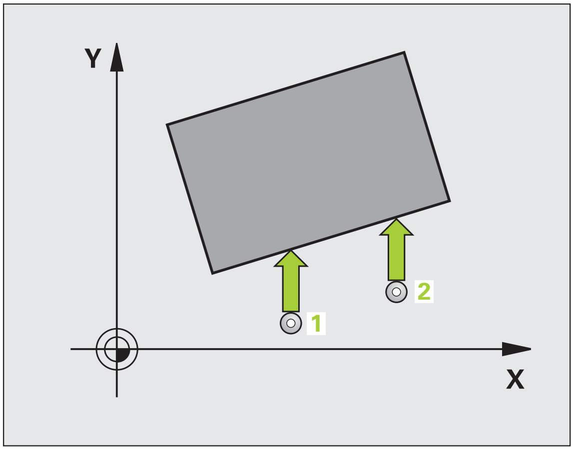
For Cycles 400, 401 and 402 you can define through parameter Q307 **`Default setting for basic rotation`** whether the measurement result is to be corrected by a known angle α (see figure at right). This enables you to measure the basic rotation against any straight line **1** of the workpiece and to establish the reference to the actual 0° direction **2**.



14.2 BASIC ROTATION (Cycle 400, DIN/ISO: G400)

Cycle run

Touch probe cycle 400 determines a workpiece misalignment by measuring two points, which must lie on a straight surface. With the basic rotation function the TNC compensates the measured value.



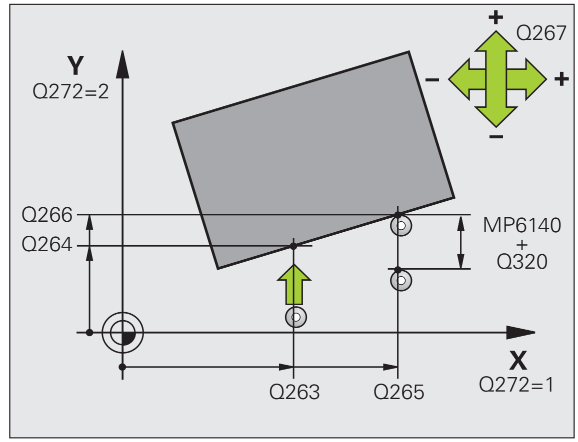
1 Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe to the programmed probe starting point **1** at rapid traverse (value from MP6150). The TNC offsets the touch probe by the set-up clearance in the direction opposite to the defined traverse direction.
2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120).
3 Then the touch probe moves to the next starting position **2** and probes the second touch point.
4 The TNC returns the touch probe to the clearance height and performs the basic rotation.

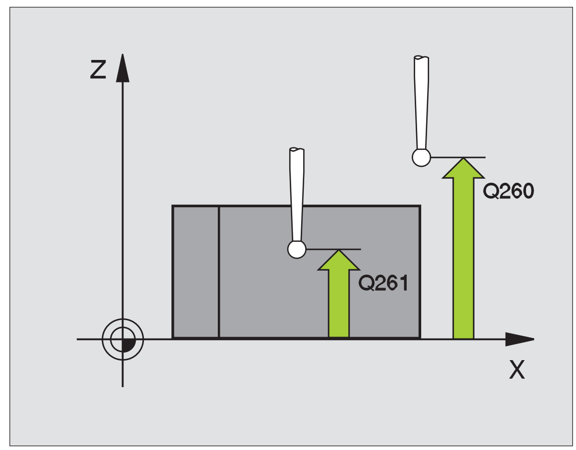
Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

The TNC will reset an active basic rotation at the beginning of the cycle.

Cycle parameters





- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 1st axis** Q265 (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 2nd axis** Q266 (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Measuring axis** Q272: Axis in the working plane in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
- **Traverse direction 1** Q267: Direction in which the touch probe is to approach the workpiece:
  –1: Negative traverse direction
  +1: Positive traverse direction
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **PREDEF**

- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
  Alternative: **PREDEF**
- **Default setting for basic rotation** Q307 (absolute): If the misalignment is to be measured against a straight line other than the reference axis, enter the angle of this reference line. The TNC will then calculate the difference between the value measured and the angle of the reference line for the basic rotation. Input range -360.000 to 360.000
- **Preset number in table** Q305: Enter the preset number in the table in which the TNC is to save the determined basic rotation. If you enter Q305=0, the TNC automatically places the determined basic rotation in the ROT menu of the Manual Operation mode. Input range 0 to 99999

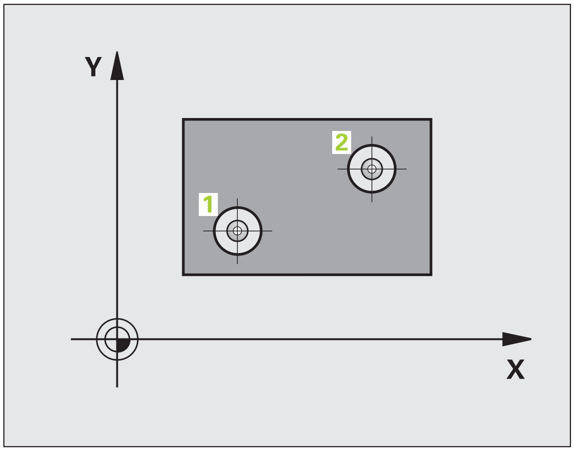
Example: NC blocks

```
5 TCH PROBE 400 BASIC ROTATION
    Q263=+10  ;1ST POINT 1ST AXIS
    Q264=+3.5 ;1ST POINT 2ND AXIS
    Q265=+25  ;2ND POINT 1ST AXIS
    Q266=+8   ;2ND POINT 2ND AXIS
    Q272=2    ;MEASURING AXIS
    Q267=+1   ;TRAVERSE DIRECTION
    Q261=-5   ;MEASURING HEIGHT
    Q320=0    ;SET-UP CLEARANCE
    Q260=+20  ;CLEARANCE HEIGHT
    Q301=0    ;MOVE TO CLEARANCE
    Q307=0    ;PRESET BASIC ROTATION
    Q305=0    ;NO. IN TABLE
```

14.3 BASIC ROTATION from Two Holes (Cycle 401, DIN/ISO: G401)

Cycle run

The Touch Probe Cycle 401 measures the centers of two holes. Then the TNC calculates the angle between the reference axis in the working plane and the line connecting the hole centers. With the basic rotation function, the TNC compensates the calculated value. As an alternative, you can also compensate the determined misalignment by rotating the rotary table.



1. Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe at rapid traverse (value from MP6150) to the point entered as center of the first hole **1**
2. Then the touch probe moves to the entered measuring height and probes four points to find the first hole center.
3. The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
4. The TNC moves the touch probe to the entered measuring height and probes four points to find the second hole center.
5. Then the TNC returns the touch probe to the clearance height and performs the basic rotation.

Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

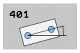
The TNC will reset an active basic rotation at the beginning of the cycle.

This touch probe cycle is not allowed when the tilted working plane function is active.

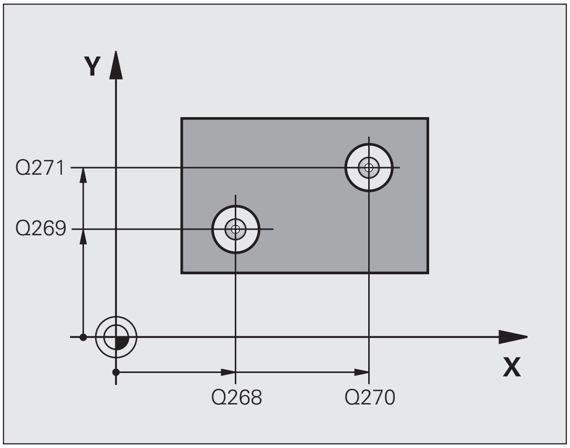
If you want to compensate the misalignment by rotating the rotary table, the TNC will automatically use the following rotary axes:

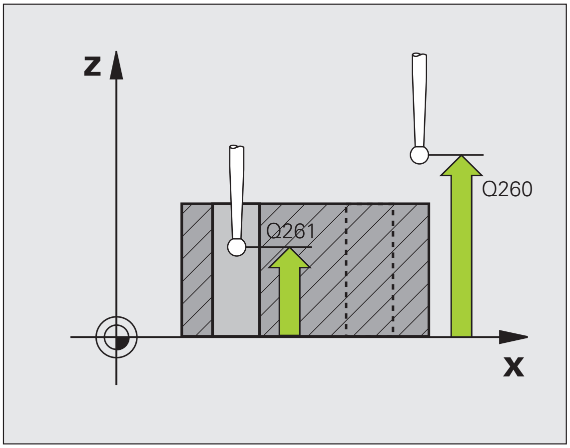
- C for tool axis Z
- B for tool axis Y
- A for tool axis X

Cycle parameters



- **1st hole: Center in 1st axis** Q268 (absolute): Center of the first hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st hole: Center in 2nd axis** Q269 (absolute): Center of the first hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Second hole: Center in 1st axis** Q270 (absolute): Center of the second hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Second hole: Center in 2nd axis** Q271 (absolute): Center of the second hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **PREDEF**
- **Default setting for basic rotation** Q307 (absolute): If the misalignment is to be measured against a straight line other than the reference axis, enter the angle of this reference line. The TNC will then calculate the difference between the value measured and the angle of the reference line for the basic rotation. Input range -360.000 to 360.000





- **Preset number in table** Q305: Enter the preset number in the table in which the TNC is to save the determined basic rotation. If you enter Q305=0, the TNC automatically places the determined basic rotation in the ROT menu of the Manual Operation mode. The parameter has no effect if the misalignment is to be compensated by a rotation of the rotary table (**Q402=1**). In this case the misalignment is not saved as an angular value. Input range 0 to 99999
- **Basic rotation / alignment** Q402: Specify whether the TNC should compensate misalignment with a basic rotation, or by rotating the rotary table:
  0: Set basic rotation
  1: Rotate the rotary table
  When you select rotary table, the TNC does not save the measured misalignment, not even when you have defined a table line in parameter **Q305.**
- **Set to zero after alignment** Q337: Definition of whether the TNC should set the display of the aligned rotary axis to zero:
  0: Do not reset the display of the rotary axis to 0 after alignment
  1: Reset the display of the rotary axis to 0 after alignment
  The TNC sets the display to 0 only if you have defined **Q402=1.**

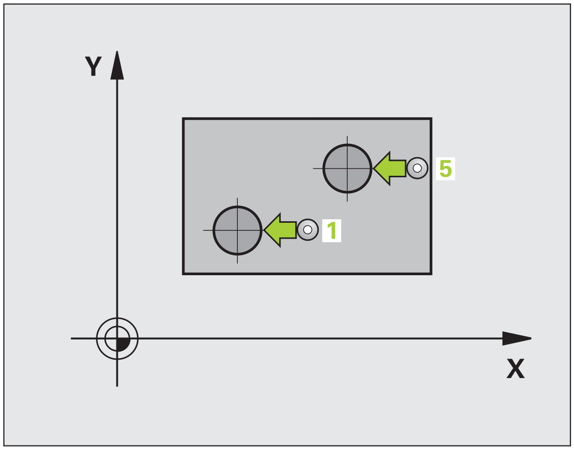
Example: NC blocks

```
5 TCH PROBE 401 ROT OF 2 HOLES
    Q268=+37  ;1ST CENTER IN 1ST AXIS
    Q269=+12  ;1ST CENTER IN 2ND AXIS
    Q270=+75  ;2ND CENTER IN 1ST AXIS
    Q271=+20  ;2ND CENTER IN 2ND AXIS
    Q261=-5   ;MEASURING HEIGHT
    Q260=+20  ;CLEARANCE HEIGHT
    Q307=0    ;PRESET BASIC ROTATION
    Q305=0    ;NO. IN TABLE
    Q402=0    ;ALIGNMENT
    Q337=0    ;SET TO ZERO
```

14.4 BASIC ROTATION over Two Studs (Cycle 402, DIN/ISO: G402)

Cycle run

The Touch Probe Cycle 402 measures the centers of two studs. Then the TNC calculates the angle between the reference axis in the working plane and the line connecting the stud centers. With the basic rotation function, the TNC compensates the calculated value. As an alternative, you can also compensate the determined misalignment by rotating the rotary table.



1 Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe in rapid traverse (value from MP6150) to the starting point **1** of the first stud.
2 Then the touch probe moves to the entered **measuring height 1** and probes four points to find the center of the first stud. The touch probe moves on a circular arc between the touch points, each of which is offset by 90°.
3 The touch probe returns to the clearance height and then positions the probe to starting point **5** of the second stud.
4 The TNC moves the touch probe to the entered **measuring height 2** and probes four points to find the center of the second stud.
5 Then the TNC returns the touch probe to the clearance height and performs the basic rotation.

Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

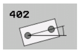
The TNC will reset an active basic rotation at the beginning of the cycle.

This touch probe cycle is not allowed when the tilted working plane function is active.

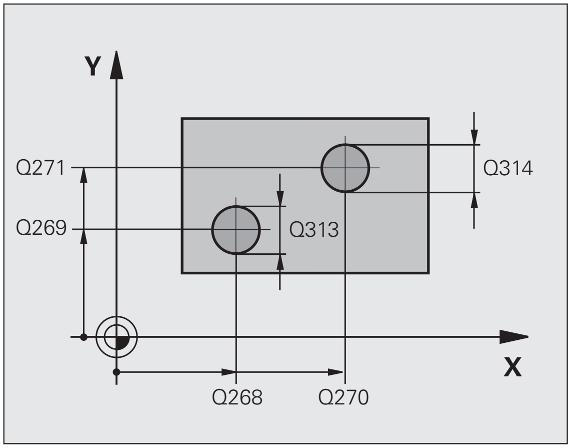
If you want to compensate the misalignment by rotating the rotary table, the TNC will automatically use the following rotary axes:

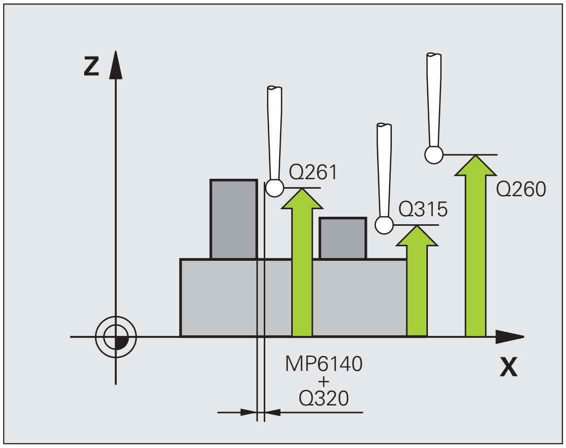
- C for tool axis Z
- B for tool axis Y
- A for tool axis X

Cycle parameters



- **`1st stud: Center in 1st axis`** (absolute): Center of the first stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **`1st stud: Center in 2nd axis`** Q269 (absolute): Center of the first stud in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **`Diameter of stud 1`** Q313: Approximate diameter of the 1st stud. Enter a value that is more likely to be too large than too small. Input range 0 to 99999.9999
- **`Measuring height 1 in the probe axis`** Q261 (absolute): Coordinate of the ball tip center (= touch point in the touch probe axis) at which stud 1 is to be measured. Input range -99999.9999 to 99999.9999
- **`2nd stud: Center in 1st axis`** Q270 (absolute): Center of the second stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **`2nd stud: Center in 2nd axis`** Q271 (absolute): Center of the second stud in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **`Diameter of stud 2`** Q314: Approximate diameter of the 2nd stud. Enter a value that is more likely to be too large than too small. Input range 0 to 99999.9999
- **`Measuring height of stud 2 in the probe axis`** Q315 (absolute): Coordinate of the ball tip center (= touch point in the touch probe axis) at which stud 2 is to be measured. Input range -99999.9999 to 99999.9999
- **`Set-up clearance`** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **`PREDEF`**
- **`Clearance height`** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **`PREDEF`**





- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
  Alternative: **PREDEF**
- **Default setting for basic rotation** Q307 (absolute): If the misalignment is to be measured against a straight line other than the reference axis, enter the angle of this reference line. The TNC will then calculate the difference between the value measured and the angle of the reference line for the basic rotation. Input range -360.000 to 360.000
- **Preset number in table** Q305: Enter the preset number in the table in which the TNC is to save the determined basic rotation. If you enter Q305=0, the TNC automatically places the determined basic rotation in the ROT menu of the Manual Operation mode. The parameter has no effect if the misalignment is to be compensated by a rotation of the rotary table (**Q402=1**). In this case the misalignment is not saved as an angular value. Input range 0 to 99999
- **Basic rotation / alignment** Q402: Specify whether the TNC should compensate misalignment with a basic rotation, or by rotating the rotary table:
  0: Set basic rotation
  1: Rotate the rotary table
  When you select rotary table, the TNC does not save the measured misalignment, not even when you have defined a table line in parameter **Q305.**
- **Set to zero after alignment** Q337: Definition of whether the TNC should set the display of the aligned rotary axis to zero:
  0: Do not reset the display of the rotary axis to 0 after alignment
  1: Reset the display of the rotary axis to 0 after alignment
  The TNC sets the display to 0 only if you have defined **Q402=1.**

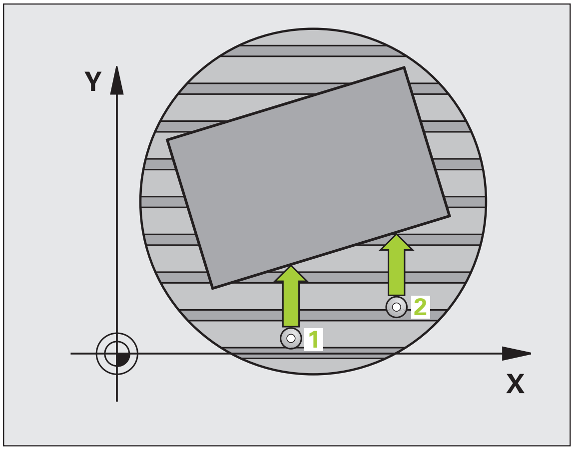
Example: NC blocks

```
5 TCH PROBE 402 ROT OF 2 STUDS
    Q268=-37  ;1ST CENTER IN 1ST AXIS
    Q269=+12  ;1ST CENTER IN 2ND AXIS
    Q313=60   ;DIAMETER OF STUD 1
    Q261=-5   ;MEASURING HEIGHT 1
    Q270=+75  ;2ND CENTER IN 1ST AXIS
    Q271=+20  ;2ND CENTER IN 2ND AXIS
    Q314=60   ;DIAMETER OF STUD 2
    Q315=-5   ;MEASURING HEIGHT 2
    Q320=0    ;SET-UP CLEARANCE
    Q260=+20  ;CLEARANCE HEIGHT
    Q301=0    ;MOVE TO CLEARANCE
    Q307=0    ;PRESET BASIC ROTATION
    Q305=0    ;NO. IN TABLE
    Q402=0    ;ALIGNMENT
    Q337=0    ;SET TO ZERO
```

14.5 BASIC ROTATION compensation via rotary axis (Cycle 403, DIN/ISO: G403)

Cycle run

Touch probe cycle 403 determines a workpiece misalignment by measuring two points, which must lie on a straight line. The TNC compensates the determined misalignment by rotating the A, B or C axis. The workpiece can be clamped in any position on the rotary table.



1. Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe to the programmed probe starting point **1** at rapid traverse (value from MP6150). The TNC offsets the touch probe by the set-up clearance in the direction opposite to the defined traverse direction.
2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120).
3. Then the touch probe moves to the next starting position **2** and probes the second touch point.
4. The TNC returns the touch probe to the clearance height and moves the rotary axis, which was defined in the cycle, by the measured value. Optionally you can have the display set to 0 after alignment.

Please note while programming:

Danger of collision!

Ensure that the **clearance height** is sufficiently large so that no collisions can occur during the final positioning of the rotary axis.

HEIDENHAIN recommends always defining the value 0 for the **Q312 Axis for compensation motion** parameter. This way the cycle automatically determines the rotary axis to be aligned, thus ensuring that the correct rotary axis is used for alignment. If Q312=0, the TNC uses the sequence of the probing points to calculate an angle with the actual direction. The angle determined goes from the first to the second probing point. If you select the A, B or C axis as compensation axis in parameter **Q312**, the cycle determines the angle, regardless of the sequence of the probing points. The calculated angle lies in the range from –90° to +90°.

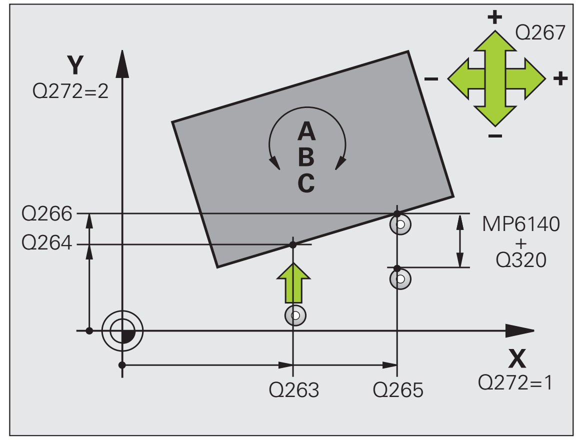
After alignment, always check the position of the rotary axis.

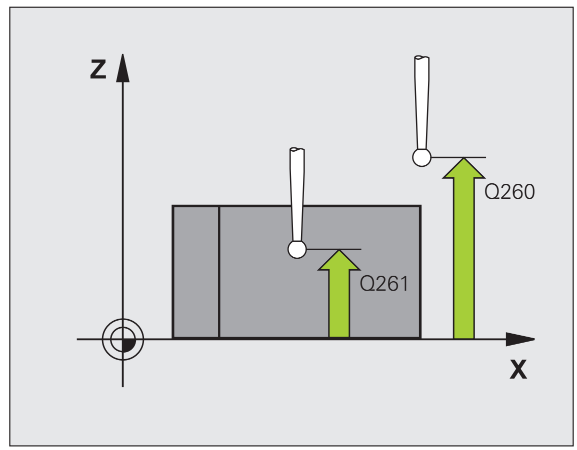
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

The TNC stores the measured angle in parameter **Q150**.

A kinematics description must be stored in the TNC in order for the compensation axis to be determined automatically by the cycle.

Cycle parameters





- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 1st axis** Q265 (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 2nd axis** Q266 (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Measuring axis** Q272: Axis in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
  3: Touch probe axis = measuring axis
- **Traverse direction 1** Q267: Direction in which the touch probe is to approach the workpiece:
  –1: Negative traverse direction
  +1: Positive traverse direction
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **PREDEF**

- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **PREDEF**
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Axis for compensation motion** Q312: Assignment of the rotary axis in which the TNC is to compensate the measured misalignment.
  0: Automatic mode – the TNC uses the active kinematics to determine the rotary axis to be aligned. In Automatic mode the first rotary axis of the table (as viewed from the workpiece) is used as compensation axis. This is the recommended setting.
  4: Compensate misalignment with rotary axis A
  5: Compensate misalignment with rotary axis B
  6: Compensate misalignment with rotary axis C
- **Set to zero after alignment** Q337: Definition of whether the TNC should set the display of the aligned rotary axis to zero:
  0: Do not reset the display of the rotary axis to 0 after alignment
  1: Reset the display of the rotary axis to 0 after alignment
- **Number in table** Q305: Enter the number in the preset table/datum table in which the TNC is to set the rotary axis to zero. Only effective if Q337 is set to 1. Input range 0 to 99999
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined angle is to be saved in the datum table or in the preset table:
  0: Write the measured angle as a datum shift in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write the measured angle in the preset table. The reference system is the machine coordinate system (REF system).
- **Reference angle? (0=ref. axis)** Q380: Angle with which the TNC is to align the probed straight line. Only effective if the rotary axis = Automatic mode or C is selected (Q312 = 0 or 6). Input range -360.000 to 360.000

Example: NC blocks

```
5 TCH PROBE 403 ROT IN C-AXIS
    Q263=+25  ;1ST POINT 1ST AXIS
    Q264=+10  ;1ST POINT 2ND AXIS
    Q265=+40  ;2ND POINT 1ST AXIS
    Q266=+17  ;2ND POINT 2ND AXIS
    Q272=2    ;MEASURING AXIS
    Q267=+1   ;TRAVERSE DIRECTION
    Q261=-5   ;MEASURING HEIGHT
    Q320=0    ;SET-UP CLEARANCE
    Q260=+20  ;CLEARANCE HEIGHT
    Q301=0    ;MOVE TO CLEARANCE
    Q312=0    ;COMPENSATION AXIS
    Q337=0    ;SET TO ZERO
    Q305=1    ;NO. IN TABLE
    Q303=+1   ;MEAS. VALUE TRANSFER
    Q380=+0   ;REFERENCE ANGLE
```

14.6 SET BASIC ROTATION (Cycle 404, DIN/ISO: G404)

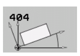
Cycle run

With Touch Probe Cycle 404, you can set any basic rotation automatically during program run. This cycle is intended primarily for resetting a previous basic rotation.

Example: NC blocks

```
5 TCH PROBE 404 BASIC ROTATION
    Q307=+0   ;PRESET BASIC ROTATION
    Q305=1    ;NO. IN TABLE
```

Cycle parameters



- **Preset value for basic rotation**: Angular value at which the basic rotation is to be set. Input range -360.000 to 360.000
- **Number in table** Q305: Enter the number in the preset/datum table in which the TNC is to save the defined basic rotation.
  -1: The TNC overwrites the active datum and activates it.
  0: The TNC copies the active datum to the datum 0, writes the basic rotation, and activates the datum 0.
  >0:The TNC only writes the defined basic rotation to the indicated datum number, but does not activate this datum. Use Cycle 247, if necessary (see "DATUM SETTING (Cycle 247, DIN/ISO: G247)" on page 288)
  Input range 0 to 99999

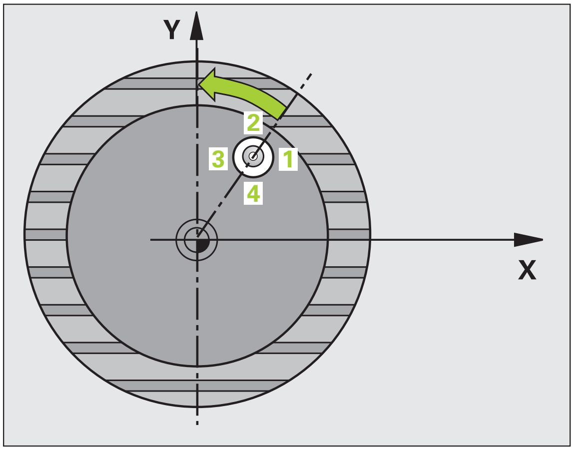
14.7 Compensating workpiece misalignment by rotating the c axis (Cycle 405, DIN/ISO: G405)

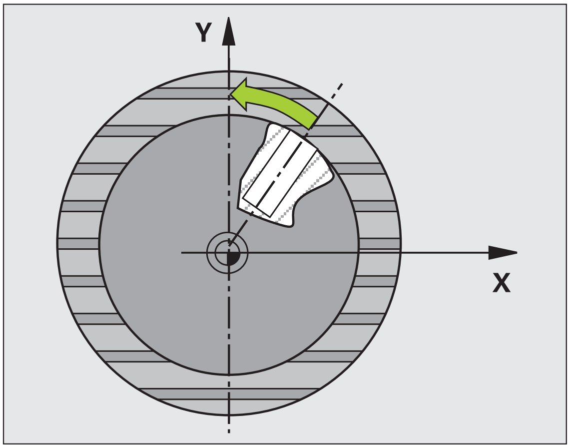
Cycle run

With Touch Probe Cycle 405, you can measure

- the angular offset between the positive Y axis of the active coordinate system and the center of a hole, or
- the angular offset between the nominal position and the actual position of a hole center.

The TNC compensates the determined angular offset by rotating the C axis. The workpiece can be clamped in any position on the rotary table, but the Y coordinate of the hole must be positive. If you measure the angular misalignment of the hole with touch probe axis Y (horizontal position of the hole), it may be necessary to execute the cycle more than once because the measuring strategy causes an inaccuracy of approx. 1% of the misalignment.





1 Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe to the probe starting point **1** at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.
2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120). The TNC derives the probing direction automatically from the programmed starting angle.
3 Then the touch probe moves in a circular arc either at measuring height or at clearance height to the next starting point **2** and probes the second touch point.
4 The TNC positions the touch probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points, and then positions the touch probe to the center of the measured hole.
5 Finally the TNC returns the touch probe to the clearance height and aligns the workpiece by rotating the table. The TNC rotates the rotary table so that the hole center after compensation lies in the direction of the positive Y axis, or at the nominal position of the hole center—both with a vertical and a horizontal touch probe axis. The measured angular misalignment is also available in parameter Q150.

Please note while programming:

Danger of collision!

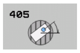
To prevent a collision between the touch probe and the workpiece, enter a **low** estimate for the nominal diameter of the pocket (or hole).

If the dimensions of the pocket and the safety clearance do not permit pre-positioning in the proximity of the touch points, the TNC always starts probing from the center of the pocket. In this case the touch probe does not return to the clearance height between the four measuring points.

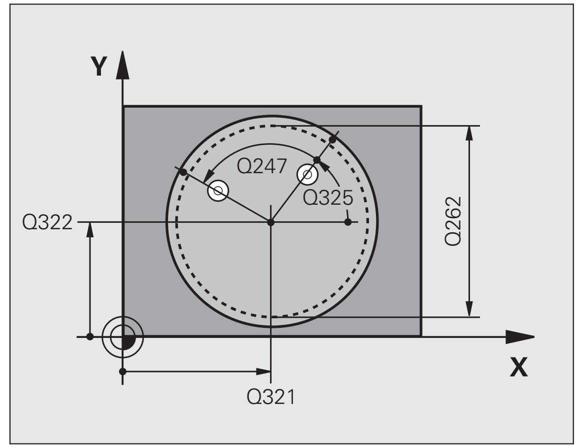
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

The smaller the angle, the less accurately the TNC can calculate the circle center. Minimum input value: 5°.

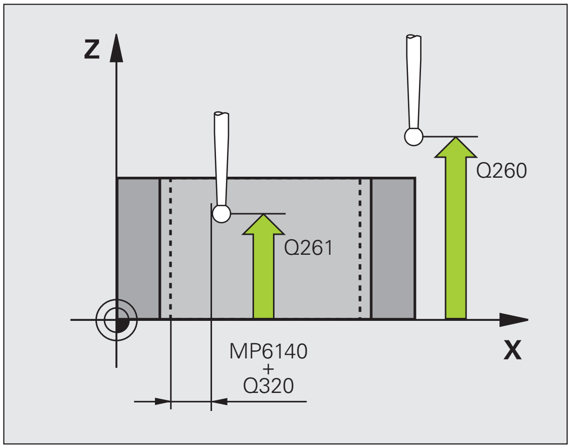
Cycle parameters



- **Center in 1st axis** Q321 (absolute): Center of the hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q322 (absolute value): Center of the hole in the minor axis of the working plane. If you program Q322 = 0, the TNC aligns the hole center to the positive Y axis. If you program Q322 not equal to 0, then the TNC aligns the hole center to the nominal position (angle of the hole center). Input range -99999.9999 to 99999.9999
- **Nominal diameter** Q262: Approximate diameter of the circular pocket (or hole). Enter a value that is more likely to be too small than too large. Input range 0 to 99999.9999
- **Starting angle** Q325 (absolute): Angle between the reference axis of the working plane and the first touch point. Input range -360.000 to 360.000
- **Stepping angle** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range -120.000 to 120.000



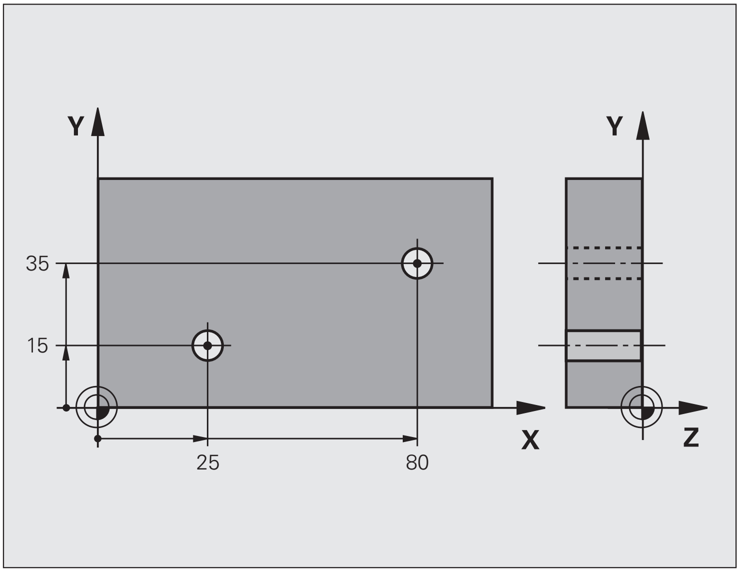
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **PREDEF**
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
  Alternative: **PREDEF**
- **Set to zero after alignment** Q337: Definition of whether the TNC should set the display of the C axis to zero, or write the angular misalignment in column C of the datum table:
  0: Set the display of the C axis to zero and write the value in line 0 of the datum table
  >0:Write the measured angular misalignment with correct algebraic signs in the datum table. Line number = value of Q337. If a C-axis shift is registered in the datum table, the TNC adds the measured angular misalignment.



Example: NC blocks

```
5 TCH PROBE 405 ROT IN C AXIS
    Q321=+50  ;CENTER IN 1ST AXIS
    Q322=+50  ;CENTER IN 2ND AXIS
    Q262=10   ;NOMINAL DIAMETER
    Q325=+0   ;STARTING ANGLE
    Q247=90   ;STEPPING ANGLE
    Q261=-5   ;MEASURING HEIGHT
    Q320=0    ;SET-UP CLEARANCE
    Q260=+20  ;CLEARANCE HEIGHT
    Q301=0    ;MOVE TO CLEARANCE
    Q337=0    ;SET TO ZERO
```

Example: Determining a basic rotation from two holes



0 BEGIN PGM CYC401 MM	
1 TOOL CALL 69 Z	
2 TCH PROBE 401 ROT OF 2 HOLES	
Q268=+25 ;1ST CENTER IN 1ST AXIS	Center of the 1st hole: X coordinate
Q269=+15 ;1ST CENTER IN 2ND AXIS	Center of the 1st hole: Y coordinate
Q270=+80 ;2ND CENTER IN 1ST AXIS	Center of the 2nd hole: X coordinate
Q271=+35 ;2ND CENTER IN 2ND AXIS	Center of the 2nd hole: Y coordinate
Q261=-5 ;MEASURING HEIGHT	Coordinate in the touch probe axis in which the measurement is made
Q260=+20 ;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision
Q307=+0 ;PRESET BASIC ROTATION	Angle of the reference line
Q402=1 ;ALIGNMENT	Compensate misalignment by rotating the rotary table
Q337=1 ;SET TO ZERO	Set the display to zero after the alignment
3 CALL PGM 35K47	Call part program
4 END PGM CYC401 MM	

15

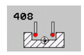
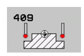
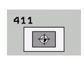
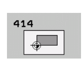
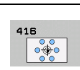
Touch Probe Cycles: Automatic Datum Setting

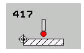
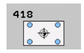
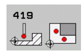
15.1 Fundamentals

Overview

The TNC offers twelve cycles for automatically finding reference points and handling them as follows:

- Setting the determined values directly as display values
- Entering the determined values in the preset table
- Entering the determined values in a datum table

Cycle	Soft key	Page
408 SLOT CENTER REF PT. Measuring the inside width of a slot, and defining the slot center as datum		Page 363
409 RIDGE CENTER REF PT. Measuring the outside width of a ridge, and defining the ridge center as datum		Page 367
410 DATUM INSIDE RECTANGLE Measuring the inside length and width of a rectangle, and defining the center as datum		Page 370
411 DATUM OUTSIDE RECTANGLE Measuring the outside length and width of a rectangle, and defining the center as datum		Page 374
412 DATUM INSIDE CIRCLE Measuring any four points on the inside of a circle, and defining the center as datum		Page 378
413 DATUM OUTSIDE CIRCLE Measuring any four points on the outside of a circle, and defining the center as datum		Page 382
414 DATUM OUTSIDE CORNER Measuring two lines from the outside of a corner, and defining the intersection as datum		Page 386
415 DATUM INSIDE CORNER Measuring two lines from the inside of a corner, and defining the intersection as datum		Page 391
416 DATUM CIRCLE CENTER (2nd soft-key row) Measuring any three holes on a bolt hole circle, and defining the bolt-hole center as datum		Page 395

Cycle	Soft key	Page
417 DATUM IN TS AXIS (2nd soft-key row) Measuring any position in the touch probe axis and defining it as datum		Page 399
418 DATUM FROM 4 HOLES (2nd soft-key row) Measuring 4 holes crosswise and defining the intersection of the lines between them as datum		Page 401
419 DATUM IN ONE AXIS (2nd soft-key row) Measuring any position in any axis and defining it as datum		Page 405

Characteristics common to all touch probe cycles for datum setting

You can also run the Touch Probe Cycles 408 to 419 during an active rotation (basic rotation or Cycle 10).

Datum point and touch probe axis

From the touch probe axis that you have defined in the measuring program the TNC determines the working plane for the datum.:

Active touch probe axis	Datum setting in
Z or W	X and Y
Y or V	Z and X
X or U	Y and Z

Saving the calculated datum

In all cycles for datum setting you can use the input parameters Q303 and Q305 to define how the TNC is to save the calculated datum:

- **Q305 = 0, Q303 = any value**
  The TNC sets the calculated datum in the display. The new datum is active immediately. At the same time, the TNC saves the datum set in the display by the cycle in line 0 of the preset table.
- **Q305 not equal to 0, Q303 = -1**

This combination can only occur if you

- read in programs containing Cycles 410 to 418 created on a TNC 4xx
- read in programs containing Cycles 410 to 418 created with an older software version on an iTNC 530
- did not specifically define the measured-value transfer with parameter Q303 when defining the cycle.

In these cases the TNC outputs an error message, since the complete handling of REF-referenced datum tables has changed. You must define a measured-value transfer yourself with parameter Q303.

- **Q305 not equal to 0, Q303 = 0**
  The TNC writes the calculated datum in the active datum table. The reference system is the active workpiece coordinate system. The value of parameter Q305 determines the datum number. **Activate the datum with Cycle 7 in the part program.**
- **Q305 not equal to 0, Q303 = 1**
  The TNC writes the calculated datum in the preset table. The reference system is the machine coordinate system (REF coordinates). The value of parameter Q305 determines the preset number. **Activate the preset with Cycle 247 in the part program.**

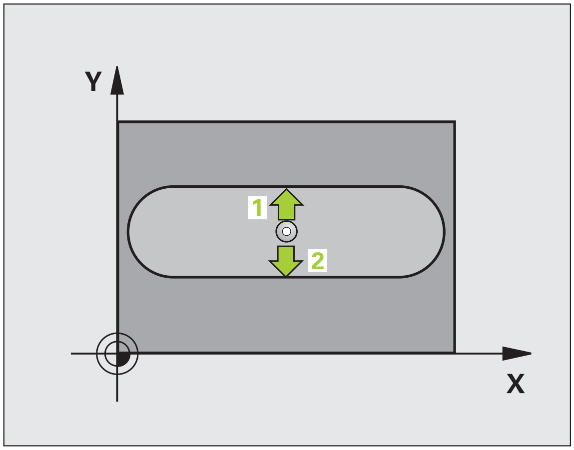
Measurement results in Q parameters

The TNC saves the measurement results of the respective touch probe cycle in the globally effective Q parameters Q150 to Q160. You can use these parameters in your program. Note the table of result parameters listed with every cycle description.

15.2 SLOT CENTER REF PT (Cycle 408, DIN/ISO: G408, FCL 3 function)

Cycle run

Touch Probe Cycle 408 finds the center of a slot and defines its center as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.



1. Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe to the probe starting point **1** at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.
2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120).
3. Then the touch probe moves either paraxially at the measuring height or linearly at the clearance height to the next starting point **2** and probes the second touch point.
4. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Saving the calculated datum" on page 362) and saves the actual values in the Q parameters listed below.
5. If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

Parameter number	Meaning
Q166	Actual value of measured slot width
Q157	Actual value of the centerline

Please note while programming:

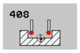
Danger of collision!

To prevent a collision between touch probe and workpiece, enter a **low** estimate for the slot width.

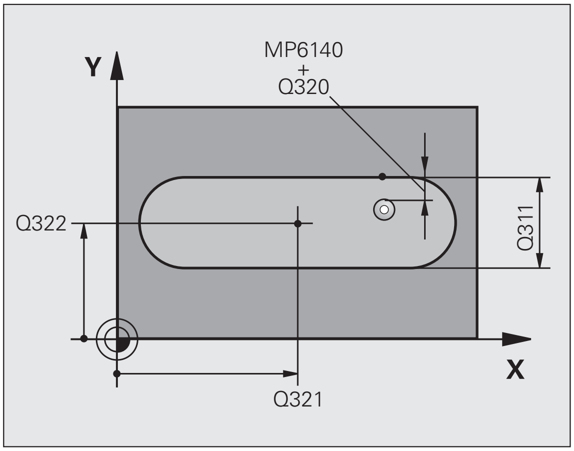
If the slot width and the safety clearance do not permit pre-positioning in the proximity of the touch points, the TNC always starts probing from the center of the slot. In this case the touch probe does not return to the clearance height between the two measuring points.

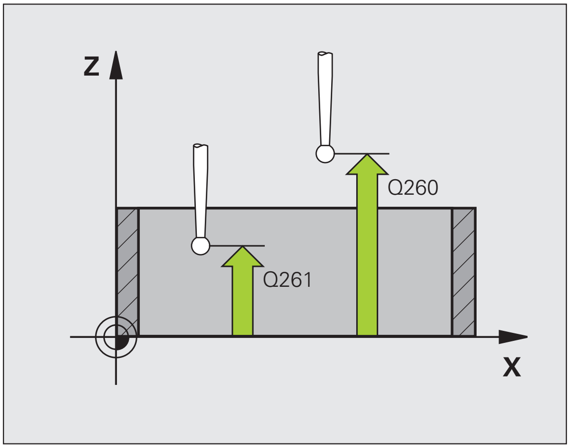
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **`Center in 1st axis`** Q321 (absolute): Center of the slot in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **`Center in 2nd axis`** Q322 (absolute): Center of the slot in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **`Width of slot`** Q311 (incremental): Width of the slot, regardless of its position in the working plane. Input range 0 to 99999.9999
- **`Measuring axis (1=1st axis / 2=2nd axis)`** Q272: Axis in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
- **`Measuring height in the touch probe axis`** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **`Set-up clearance`** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **`PREDEF`**
- **`Clearance height`** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **`PREDEF`**





- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
  Alternative: **PREDEF**
- **Number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the slot center. If you enter Q305=0 and Q303=1, the TNC automatically sets the display so that the new datum is on the slot center. If you enter Q305=0 and Q303=0, the TNC writes the slot center to line 0 of the datum table. Input range 0 to 99999
- **New datum** Q405 (absolute): Coordinate in the measuring axis at which the TNC should set the calculated slot center. Default setting = 0. Input range -99999.9999 to 99999.9999
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).

- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set datum in the touch probe axis
  1: Set the datum in the touch probe axis
- **Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999

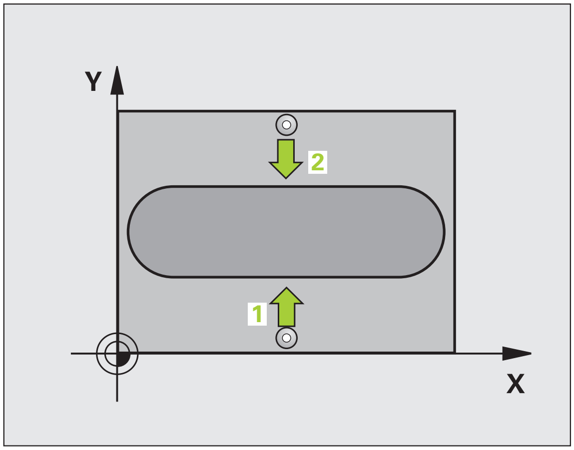
Example: NC blocks

```
5 TCH PROBE 408 SLOT CENTER REF PT
    Q321=+50  ;CENTER IN 1ST AXIS
    Q322=+50  ;CENTER IN 2ND AXIS
    Q311=25   ;SLOT WIDTH
    Q272=1    ;MEASURING AXIS
    Q261=-5   ;MEASURING HEIGHT
    Q320=0    ;SET-UP CLEARANCE
    Q260=+20  ;CLEARANCE HEIGHT
    Q301=0    ;MOVE TO CLEARANCE
    Q305=10   ;NO. IN TABLE
    Q405=+0   ;DATUM
    Q303=+1   ;MEAS. VALUE TRANSFER
    Q381=1    ;PROBE IN TS AXIS
    Q382=+85  ;1ST CO. FOR TS AXIS
    Q383=+50  ;2ND CO. FOR TS AXIS
    Q384=+0   ;3RD CO. FOR TS AXIS
    Q333=+1   ;DATUM
```

15.3 RIDGE CENTER REF PT (Cycle 409, DIN/ISO: G409, FCL 3 function)

Cycle run

Touch Probe Cycle 409 finds the center of a ridge and defines its center as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.



1 Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe to the probe starting point **1** at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.
2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120).
3 Then the touch probe moves at clearance height to the next touch point **2** and probes the second touch point.
4 Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Saving the calculated datum" on page 362) and saves the actual values in the Q parameters listed below.
5 If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

Parameter number	Meaning
Q166	Actual value of measured ridge width
Q157	Actual value of the centerline

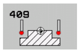
Please note while programming:

Danger of collision!

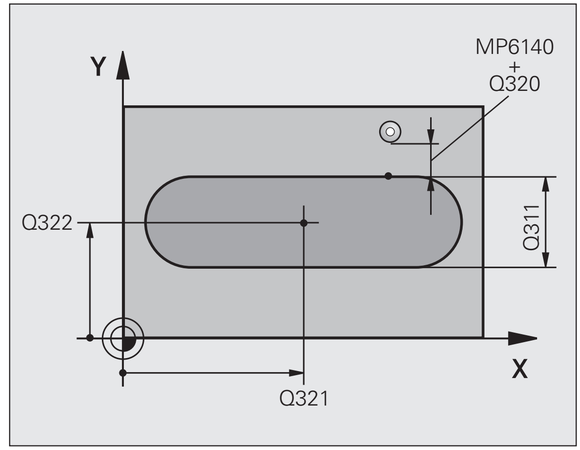
To prevent a collision between touch probe and workpiece, enter a **high** estimate for the ridge width.

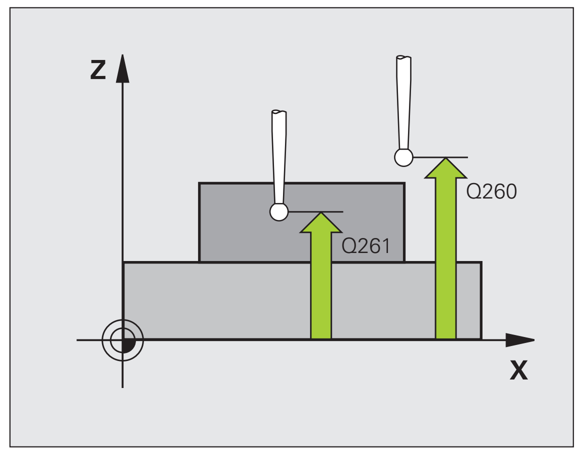
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **`Center in 1st axis`** Q321 (absolute): Center of the ridge in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **`Center in 2nd axis`** Q322 (absolute): Center of the ridge in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **`Width of ridge`** Q311 (incremental): Width of the ridge, regardless of its position in the working plane. Input range 0 to 99999.9999
- **`Measuring axis (1=1st axis / 2=2nd axis)`** Q272: Axis in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
- **`Measuring height in the touch probe axis`** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **`Set-up clearance`** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **`PREDEF`**
- **`Clearance height`** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **`PREDEF`**
- **`Number in table`** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the ridge center. If you enter Q305=0 and Q303=1, the TNC automatically sets the display so that the new datum is on the ridge center. If you enter Q305=0 and Q303=0, the TNC writes the ridge center to line 0 of the datum table. Input range 0 to 99999
- **`New datum`** Q405 (absolute): Coordinate in the measuring axis at which the TNC should set the calculated ridge center. Default setting = 0. Input range -99999.9999 to 99999.9999





- **`Measured-value transfer (0, 1)`** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).
- **`Probe in TS axis`** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set datum in the touch probe axis
  1: Set the datum in the touch probe axis
- **`Probe TS axis: Coord. 1st axis`** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **`Probe TS axis: Coord. 2nd axis`** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **`Probe TS axis: Coord. 3rd axis`** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **`New datum in TS axis`** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999

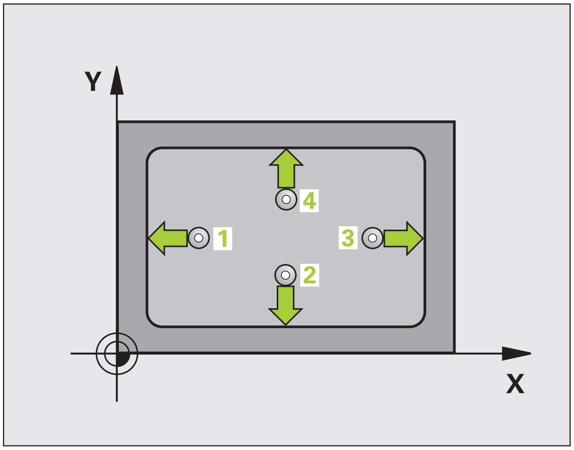
Example: NC blocks

```
5 TCH PROBE 409 SLOT CENTER RIDGE
     Q321=+50  ;CENTER IN 1ST AXIS
     Q322=+50  ;CENTER IN 2ND AXIS
     Q311=25   ;RIDGE WIDTH
     Q272=1    ;MEASURING AXIS
     Q261=-5   ;MEASURING HEIGHT
     Q320=0    ;SET-UP CLEARANCE
     Q260=+20  ;CLEARANCE HEIGHT
     Q305=10   ;NO. IN TABLE
     Q405=+0   ;DATUM
     Q303=+1   ;MEAS. VALUE TRANSFER
     Q381=1    ;PROBE IN TS AXIS
     Q382=+85  ;1ST CO. FOR TS AXIS
     Q383=+50  ;2ND CO. FOR TS AXIS
     Q384=+0   ;3RD CO. FOR TS AXIS
     Q333=+1   ;DATUM
```

15.4 DATUM FROM INSIDE OF RECTANGLE (Cycle 410, DIN/ISO: G410)

Cycle run

Touch Probe Cycle 410 finds the center of a rectangular pocket and defines its center as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.



1. Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe to the probe starting point **1** at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.
2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120).
3. Then the touch probe moves either paraxially at the measuring height or linearly at the clearance height to the next starting point **2** and probes the second touch point.
4. The TNC positions the touch probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
5. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Saving the calculated datum" on page 362).
6. If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing and saves the actual values in the following Q parameters.

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of length in the reference axis
Q155	Actual value of length in the minor axis

Please note while programming:

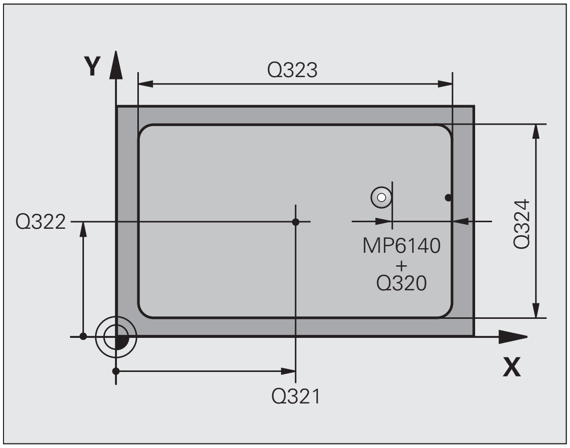
Danger of collision!

To prevent a collision between touch probe and workpiece, enter **low** estimates for the lengths of the first and second sides.

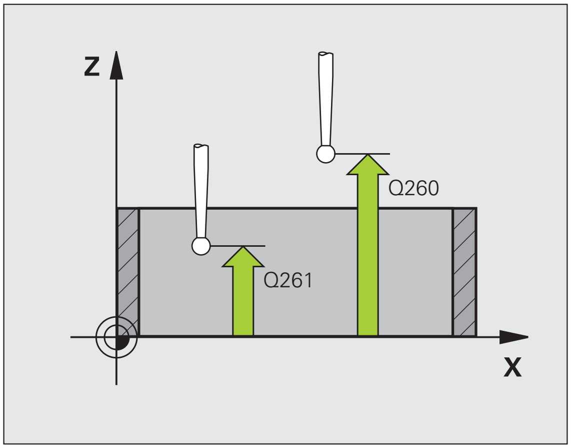
If the dimensions of the pocket and the safety clearance do not permit pre-positioning in the proximity of the touch points, the TNC always starts probing from the center of the pocket. In this case the touch probe does not return to the clearance height between the four measuring points.

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **`Center in 1st axis`** Q321 (absolute): Center of the pocket in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **`Center in 2nd axis`** Q322 (absolute): Center of the pocket in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **`1st side length`** Q323 (incremental): Pocket length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- **`2nd side length`** Q324 (incremental): Pocket length, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- **`Measuring height in the touch probe axis`** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **`Set-up clearance`** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **`PREDEF`**
- **`Clearance height`** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **`PREDEF`**



- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
  Alternative: **PREDEF**
- **Number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the pocket center. If you enter Q305=0 and Q303=1, the TNC automatically sets the display so that the new datum is at the center of the pocket. If you enter Q305=0 and Q303=0, the TNC writes the center of the pocket to line 0 of the datum table. Input range 0 to 99999
- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the pocket center. Default setting = 0. Input range -99999.9999 to 99999.9999
- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the pocket center. Default setting = 0. Input range -99999.9999 to 99999.9999
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  –1: Do not use. Is entered by the TNC when old programs are read in (see "Saving the calculated datum" on page 362).
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).

- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set datum in the touch probe axis
  1: Set the datum in the touch probe axis
- **Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999

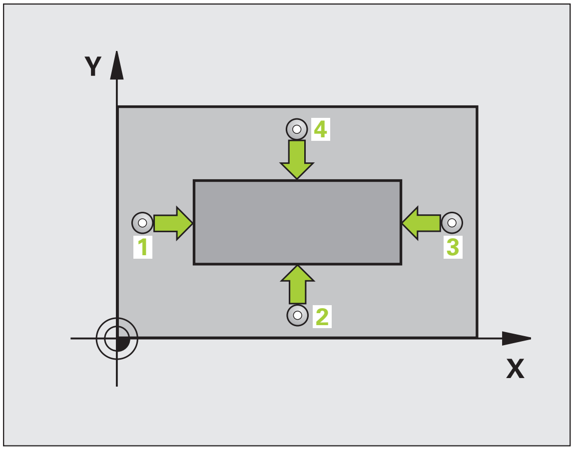
Example: NC blocks

```
5 TCH PROBE 410 DATUM INSIDE RECTAN.
    Q321=+50   ;CENTER IN 1ST AXIS
    Q322=+50   ;CENTER IN 2ND AXIS
    Q323=60    ;1ST SIDE LENGTH
    Q324=20    ;2ND SIDE LENGTH
    Q261=-5    ;MEASURING HEIGHT
    Q320=0     ;SET-UP CLEARANCE
    Q260=+20   ;CLEARANCE HEIGHT
    Q301=0     ;MOVE TO CLEARANCE
    Q305=10    ;NO. IN TABLE
    Q331=+0    ;DATUM
    Q332=+0    ;DATUM
    Q303=+1    ;MEAS. VALUE TRANSFER
    Q381=1     ;PROBE IN TS AXIS
    Q382=+85   ;1ST CO. FOR TS AXIS
    Q383=+50   ;2ND CO. FOR TS AXIS
    Q384=+0    ;3RD CO. FOR TS AXIS
    Q333=+1    ;DATUM
```

15.5 DATUM FROM OUTSIDE OF RECTANGLE (Cycle 411, DIN/ISO: G411)

Cycle run

Touch Probe Cycle 411 finds the center of a rectangular stud and defines its center as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.



1. Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe to the probe starting point **1** at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.
2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120).
3. Then the touch probe moves either paraxially at the measuring height or linearly at the clearance height to the next starting point **2** and probes the second touch point.
4. The TNC positions the touch probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
5. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Saving the calculated datum" on page 362).
6. If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing and saves the actual values in the following Q parameters.

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of length in the reference axis
Q155	Actual value of length in the minor axis

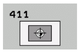
Please note while programming:

Danger of collision!

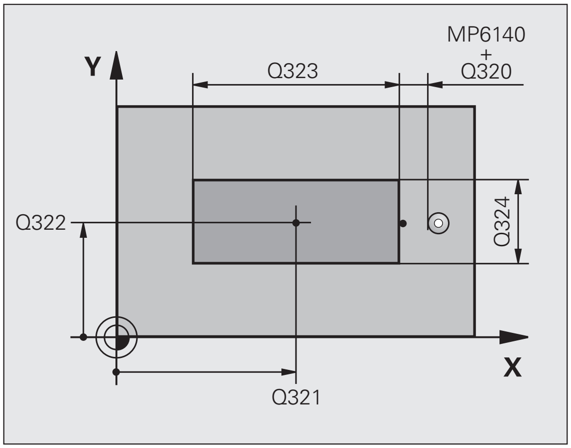
To prevent a collision between the touch probe and workpiece, enter **high** estimates for the lengths of the first and second sides.

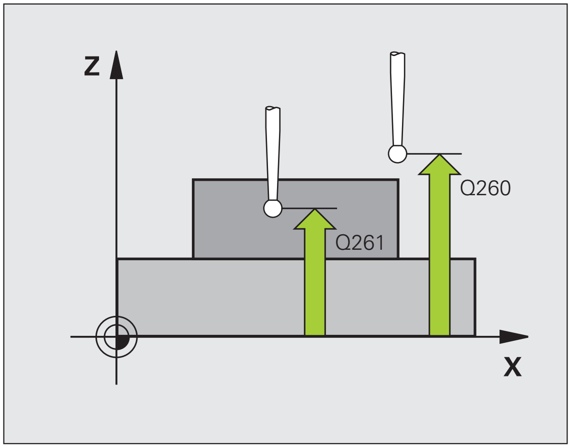
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **Center in 1st axis** Q321 (absolute): Center of the stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q322 (absolute): Center of the stud in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st side length** Q323 (incremental): Stud length, parallel to the reference axis of the working plane Input range 0 to 99999.9999
- **2nd side length** Q324 (incremental): Stud length, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **PREDEF**





- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
  Alternative: **PREDEF**
- **Number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the stud center. If you enter Q305=0 and Q303=1, the TNC automatically sets the display so that the new datum is on the stud center. If you enter Q305=0 and Q303=0, the TNC writes the center of the stud to line 0 of the datum table. Input range 0 to 99999
- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the stud center. Default setting = 0. Input range -99999.9999 to 99999.9999
- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the stud center. Default setting = 0. Input range -99999.9999 to 99999.9999
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  –1: Do not use. Is entered by the TNC when old programs are read in (see "Saving the calculated datum" on page 362).
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).

- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set datum in the touch probe axis
  1: Set the datum in the touch probe axis
- **Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999

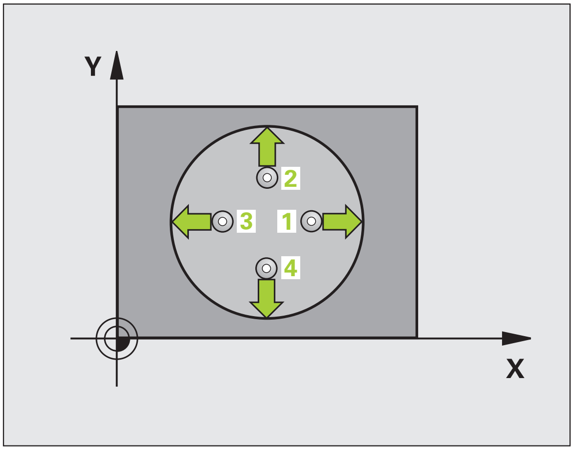
Example: NC blocks

```
5 TCH PROBE 411 DATUM OUTS. RECTAN.
    Q321=+50  ;CENTER IN 1ST AXIS
    Q322=+50  ;CENTER IN 2ND AXIS
    Q323=60   ;1ST SIDE LENGTH
    Q324=20   ;2ND SIDE LENGTH
    Q261=-5   ;MEASURING HEIGHT
    Q320=0    ;SET-UP CLEARANCE
    Q260=+20  ;CLEARANCE HEIGHT
    Q301=0    ;MOVE TO CLEARANCE
    Q305=0    ;NO. IN TABLE
    Q331=+0   ;DATUM
    Q332=+0   ;DATUM
    Q303=+1   ;MEAS. VALUE TRANSFER
    Q381=1    ;PROBE IN TS AXIS
    Q382=+85  ;1ST CO. FOR TS AXIS
    Q383=+50  ;2ND CO. FOR TS AXIS
    Q384=+0   ;3RD CO. FOR TS AXIS
    Q333=+1   ;DATUM
```

15.6 DATUM FROM INSIDE OF CIRCLE (Cycle 412, DIN/ISO: G412)

Cycle run

Touch Probe Cycle 412 finds the center of a circular pocket (or of a hole) and defines its center as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.



1. Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe to the probe starting point **1** at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.
2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120). The TNC derives the probing direction automatically from the programmed starting angle.
3. Then the touch probe moves in a circular arc either at measuring height or at clearance height to the next starting point **2** and probes the second touch point.
4. The TNC positions the touch probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
5. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Saving the calculated datum" on page 362) and saves the actual values in the Q parameters listed below.
6. If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter

Please note while programming:

Danger of collision!

To prevent a collision between the touch probe and the workpiece, enter a **low** estimate for the nominal diameter of the pocket (or hole).

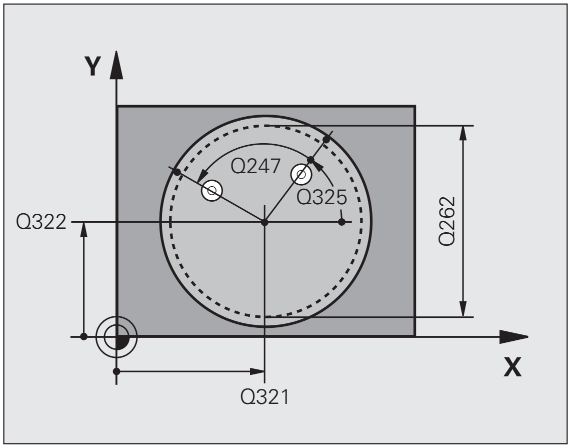
If the dimensions of the pocket and the safety clearance do not permit pre-positioning in the proximity of the touch points, the TNC always starts probing from the center of the pocket. In this case the touch probe does not return to the clearance height between the four measuring points.

The smaller the angle increment Q247, the less accurately the TNC can calculate the datum. Minimum input value: 5°

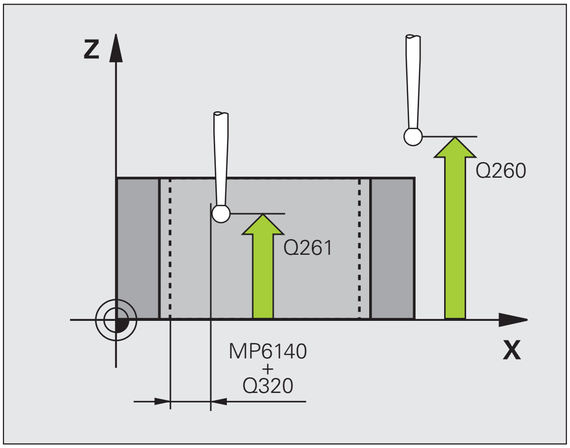
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle parameters

- **`Center in 1st axis`** Q321 (absolute): Center of the pocket in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **`Center in 2nd axis`** Q322 (absolute): Center of the pocket in the minor axis of the working plane. If you program Q322 = 0, the TNC aligns the hole center to the positive Y axis. If you program Q322 not equal to 0, then the TNC aligns the hole center to the nominal position. Input range -99999.9999 to 99999.9999
- **`Nominal diameter`** Q262: Approximate diameter of the circular pocket (or hole). Enter a value that is more likely to be too small than too large. Input range 0 to 99999.9999
- **`Starting angle`** Q325 (absolute): Angle between the reference axis of the working plane and the first touch point. Input range -360.0000 to 360.0000
- **`Stepping angle`** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range -120.0000 to 120.0000



- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **PREDEF**
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
  Alternative: **PREDEF**
- **Number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the pocket center. If you enter Q305=0 and Q303=1, the TNC automatically sets the display so that the new datum is at the center of the pocket. If you enter Q305=0 and Q303=0, the TNC writes the center of the pocket to line 0 of the datum table. Input range 0 to 99999
- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the pocket center. Default setting = 0. Input range -99999.9999 to 99999.9999
- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the pocket center. Default setting = 0. Input range -99999.9999 to 99999.9999
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  –1: Do not use. Is entered by the TNC when old programs are read in (see "Saving the calculated datum" on page 362).
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).



- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set datum in the touch probe axis
  1: Set the datum in the touch probe axis
- **Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999
- **No. of measuring points (4/3)** Q423: Specify whether the TNC should measure the hole with 4 or 3 probing points:
  4: Use 4 measuring points (standard setting)
  3: Use 3 measuring points
- **Type of traverse? Line=0/Arc=1** Q365: Definition of the path function with which the touch probe is to move between the measuring points if "traverse to clearance height" (Q301=1) is active.
  0: Move in a straight line between measuring points
  1: Move in a circular arc on the pitch circle diameter between measuring points

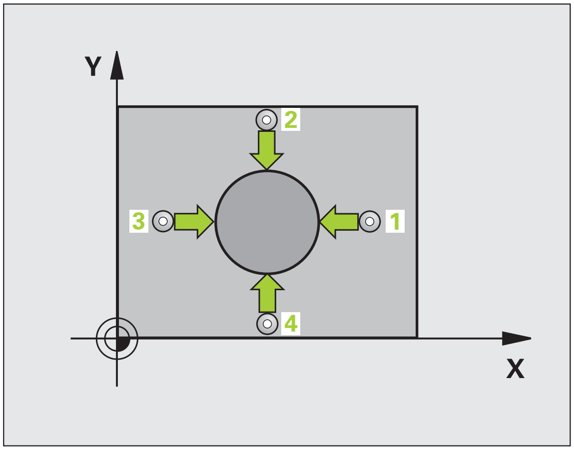
Example: NC blocks

```
5 TCH PROBE 412 DATUM INSIDE CIRCLE
    Q321=+50   ;CENTER IN 1ST AXIS
    Q322=+50   ;CENTER IN 2ND AXIS
    Q262=75    ;NOMINAL DIAMETER
    Q325=+0    ;STARTING ANGLE
    Q247=+60   ;STEPPING ANGLE
    Q261=-5    ;MEASURING HEIGHT
    Q320=0     ;SET-UP CLEARANCE
    Q260=+20   ;CLEARANCE HEIGHT
    Q301=0     ;MOVE TO CLEARANCE
    Q305=12    ;NO. IN TABLE
    Q331=+0    ;DATUM
    Q332=+0    ;DATUM
    Q303=+1    ;MEAS. VALUE TRANSFER
    Q381=1     ;PROBE IN TS AXIS
    Q382=+85   ;1ST CO. FOR TS AXIS
    Q383=+50   ;2ND CO. FOR TS AXIS
    Q384=+0    ;3RD CO. FOR TS AXIS
    Q333=+1    ;DATUM
    Q423=4     ;NO. OF MEAS. POINTS
    Q365=1     ;TYPE OF TRAVERSE
```

15.7 DATUM FROM OUTSIDE OF CIRCLE (Cycle 413, DIN/ISO: G413)

Cycle run

Touch Probe Cycle 413 finds the center of a circular stud and defines it as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.



1. Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe to the probe starting point **1** at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.
2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120). The TNC derives the probing direction automatically from the programmed starting angle.
3. Then the touch probe moves in a circular arc either at measuring height or at clearance height to the next starting point **2** and probes the second touch point.
4. The TNC positions the touch probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
5. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Saving the calculated datum" on page 362) and saves the actual values in the Q parameters listed below.
6. If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter

Please note while programming:

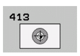
Danger of collision!

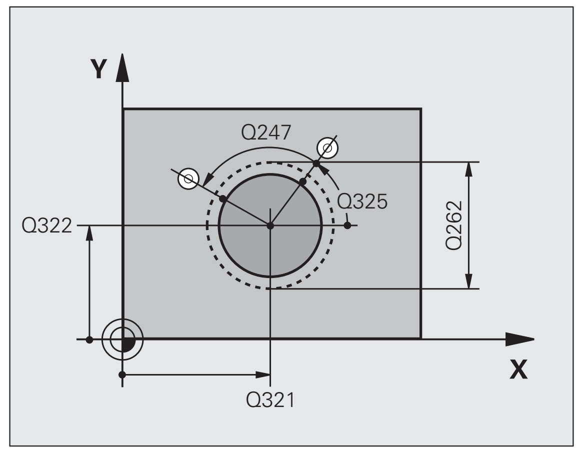
To prevent a collision between touch probe and workpiece, enter a **high** estimate for the nominal diameter of the stud.

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

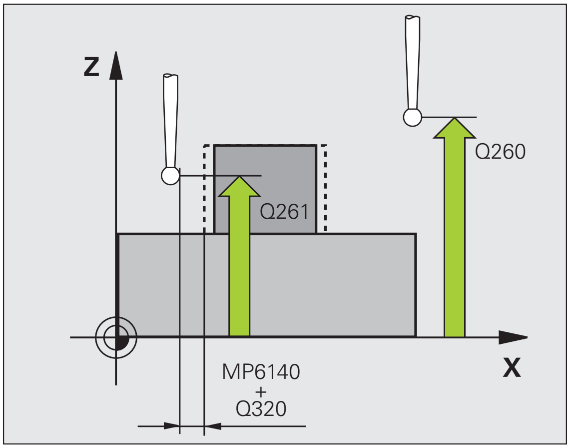
The smaller the angle increment Q247, the less accurately the TNC can calculate the datum. Minimum input value: 5°.

Cycle parameters





- **Center in 1st axis** Q321 (absolute): Center of the stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q322 (absolute): Center of the stud in the minor axis of the working plane. If you program Q322 = 0, the TNC aligns the hole center to the positive Y axis. If you program Q322 not equal to 0, then the TNC aligns the hole center to the nominal position. Input range -99999.9999 to 99999.9999
- **Nominal diameter** Q262: Approximate diameter of the stud. Enter a value that is more likely to be too large than too small. Input range 0 to 99999.9999
- **Starting angle** Q325 (absolute): Angle between the reference axis of the working plane and the first touch point. Input range -360.0000 to 360.0000
- **Stepping angle** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range -120.0000 to 120.0000



- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **PREDEF**
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
  Alternative: **PREDEF**
- **Number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the stud center. If you enter Q305=0 and Q303=1, the TNC automatically sets the display so that the new datum is on the stud center. If you enter Q305=0 and Q303=0, the TNC writes the center of the stud to line 0 of the datum table. Input range 0 to 99999
- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the stud center. Default setting = 0. Input range -99999.9999 to 99999.9999
- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the stud center. Default setting = 0. Input range -99999.9999 to 99999.9999
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  –1: Do not use. Is entered by the TNC when old programs are read in (see "Saving the calculated datum" on page 362).
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).

- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set datum in the touch probe axis
  1: Set the datum in the touch probe axis
- **Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0
- **No. of measuring points (4/3)** Q423: Specify whether the TNC should measure the stud with 4 or 3 probing points:
  4: Use 4 measuring points (standard setting)
  3: Use 3 measuring points
- **Type of traverse? Line=0/Arc=1** Q365: Definition of the path function with which the touch probe is to move between the measuring points if "traverse to clearance height" (Q301=1) is active.
  0: Move in a straight line between measuring points
  1: Move in a circular arc on the pitch circle diameter between measuring points

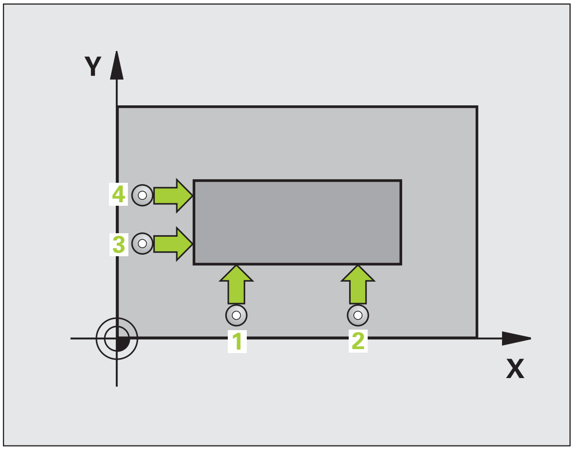
Example: NC blocks

```
5 TCH PROBE 413 DATUM OUTSIDE CIRCLE
    Q321=+50   ;CENTER IN 1ST AXIS
    Q322=+50   ;CENTER IN 2ND AXIS
    Q262=75    ;NOMINAL DIAMETER
    Q325=+0    ;STARTING ANGLE
    Q247=+60   ;STEPPING ANGLE
    Q261=-5    ;MEASURING HEIGHT
    Q320=0     ;SET-UP CLEARANCE
    Q260=+20   ;CLEARANCE HEIGHT
    Q301=0     ;MOVE TO CLEARANCE
    Q305=15    ;NO. IN TABLE
    Q331=+0    ;DATUM
    Q332=+0    ;DATUM
    Q303=+1    ;MEAS. VALUE TRANSFER
    Q381=1     ;PROBE IN TS AXIS
    Q382=+85   ;1ST CO. FOR TS AXIS
    Q383=+50   ;2ND CO. FOR TS AXIS
    Q384=+0    ;3RD CO. FOR TS AXIS
    Q333=+1    ;DATUM
    Q423=4     ;NO. OF MEAS. POINTS
    Q365=1     ;TYPE OF TRAVERSE
```

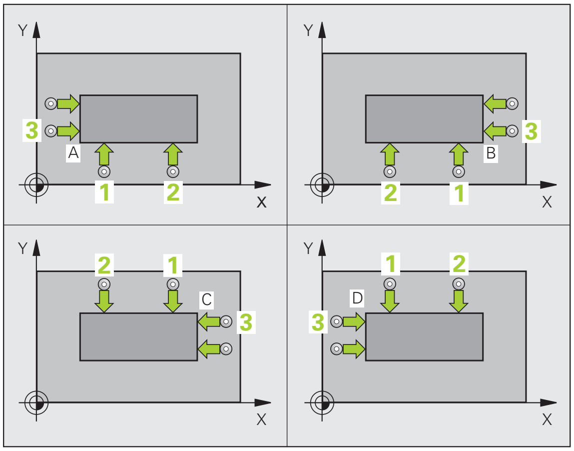
15.8 DATUM FROM OUTSIDE OF CORNER (Cycle 414, DIN/ISO: G414)

Cycle run

Touch Probe Cycle 414 finds the intersection of two lines and defines it as the datum. If desired, the TNC can also enter the intersection into a datum table or preset table.

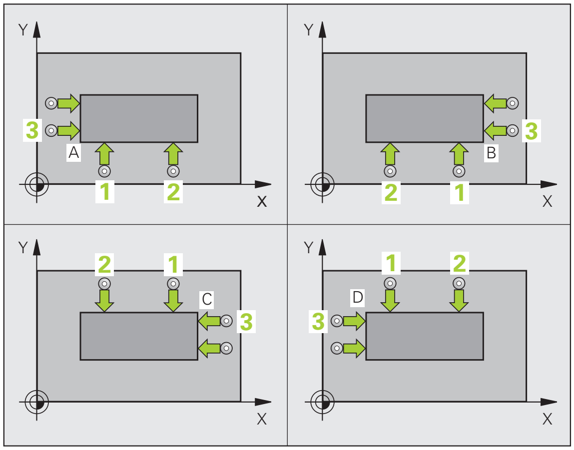


1 Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe at rapid traverse (value from MP6150) to the first touch point **1** (see figure at upper right). The TNC offsets the touch probe by the safety clearance in the direction opposite to the respective traverse direction.
2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120). The TNC derives the probing direction automatically from the programmed 3rd measuring point.
3 Then the touch probe moves to the next starting position **2** and probes the second touch point.
4 The TNC positions the touch probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
5 Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Saving the calculated datum" on page 362) and saves the coordinates of the determined corner in the Q parameters listed below.
6 If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.



Parameter number	Meaning
Q151	Actual value of corner in reference axis
Q152	Actual value of corner in minor axis

Please note while programming:



Before a cycle definition you must have programmed a tool call to define the touch probe axis.

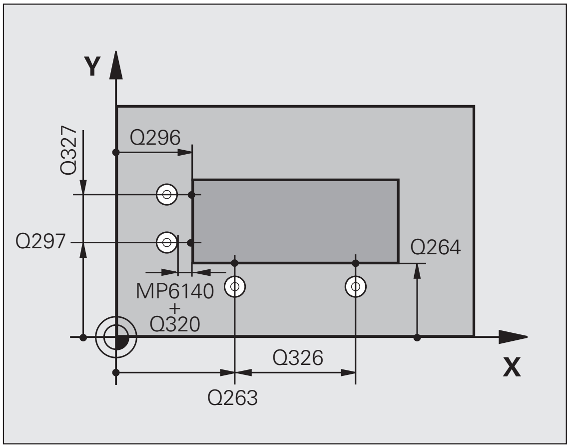
The TNC always measures the first line in the direction of the minor axis of the working plane.

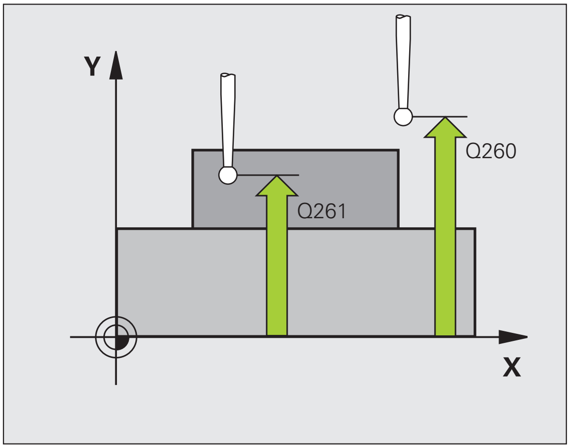
By defining the positions of the measuring points **1** and **3** you also determine the corner at which the TNC sets the datum (see figure at mid-right and table at lower right).

Corner	X coordinate	Y coordinate
A	Point **1** greater than point **3**	Point **1** less than point **3**
B	Point **1** less than point **3**	Point **1** less than point **3**
C	Point **1** less than point **3**	Point **1** greater than point **3**
D	Point **1** greater than point **3**	Point **1** greater than point **3**

Cycle parameters

- **`1st meas. point 1st axis`** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **`1st meas. point 2nd axis`** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **`Spacing in 1st axis`** Q326 (incremental): Distance between the first and second measuring points in the reference axis of the working plane. Input range 0 to 99999.9999
- **`3rd meas. point 1st axis`** Q296 (absolute): Coordinate of the third touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **`3rd meas. point 2nd axis`** Q297 (absolute): Coordinate of the third touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **`Spacing in 2nd axis`** Q327 (incremental): Distance between third and fourth measuring points in the minor axis of the working plane. Input range 0 to 99999.9999
- **`Measuring height in the touch probe axis`** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **`Set-up clearance`** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **`PREDEF`**
- **`Clearance height`** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **`PREDEF`**





- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
  Alternative: **PREDEF**
- **Execute basic rotation** Q304: Definition of whether the TNC should compensate workpiece misalignment with a basic rotation:
  0: No basic rotation
  1: Basic rotation
- **Number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the corner. If you enter Q305=0 and Q303=1, the TNC automatically sets the display so that the new datum is on the corner. If you enter Q305=0 and Q303=0, the TNC writes the corner to line 0 of the datum table. Input range 0 to 99999
- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the corner. Default setting = 0. Input range -99999.9999 to 99999.9999
- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the calculated corner. Default setting = 0. Input range -99999.9999 to 99999.9999
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  –1: Do not use. Is entered by the TNC when old programs are read in (see "Saving the calculated datum" on page 362).
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).

- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set datum in the touch probe axis
  1: Set the datum in the touch probe axis
- **Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999

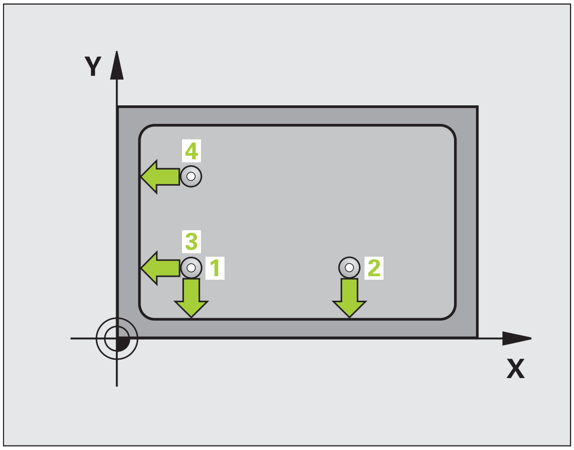
Example: NC blocks

```
5 TCH PROBE 414 DATUM OUTSIDE CORNER
    Q263=+37   ;1ST POINT 1ST AXIS
    Q264=+7    ;1ST POINT 2ND AXIS
    Q326=50    ;SPACING IN 1ST AXIS
    Q296=+95   ;3RD POINT 1ST AXIS
    Q297=+25   ;3RD POINT 2ND AXIS
    Q327=45    ;SPACING IN 2ND AXIS
    Q261=-5    ;MEASURING HEIGHT
    Q320=0     ;SET-UP CLEARANCE
    Q260=+20   ;CLEARANCE HEIGHT
    Q301=0     ;MOVE TO CLEARANCE
    Q304=0     ;BASIC ROTATION
    Q305=7     ;NO. IN TABLE
    Q331=+0    ;DATUM
    Q332=+0    ;DATUM
    Q303=+1    ;MEAS. VALUE TRANSFER
    Q381=1     ;PROBE IN TS AXIS
    Q382=+85   ;1ST CO. FOR TS AXIS
    Q383=+50   ;2ND CO. FOR TS AXIS
    Q384=+0    ;3RD CO. FOR TS AXIS
    Q333=+1    ;DATUM
```

15.9 DATUM FROM INSIDE OF CORNER (Cycle 415, DIN/ISO: G415)

Cycle run

Touch Probe Cycle 415 finds the intersection of two lines and defines it as the datum. If desired, the TNC can also enter the intersection into a datum table or preset table.



1. Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe at rapid traverse (value from MP6150) to the first touch point **1** (see figure at upper right) that you have defined in the cycle. The TNC offsets the touch probe by the safety clearance in the direction opposite to the respective traverse direction.
2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120). The probing direction is derived from the number by which you identify the corner.
3. Then the touch probe moves to the next starting position **2** and probes the second touch point.
4. The TNC positions the touch probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
5. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Saving the calculated datum" on page 362) and saves the coordinates of the determined corner in the Q parameters listed below.
6. If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

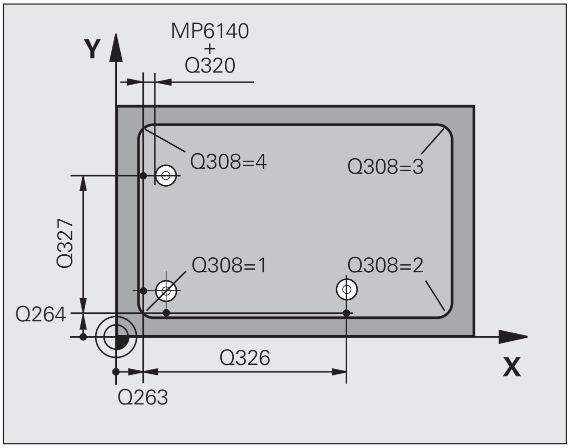
Parameter number	Meaning
Q151	Actual value of corner in reference axis
Q152	Actual value of corner in minor axis

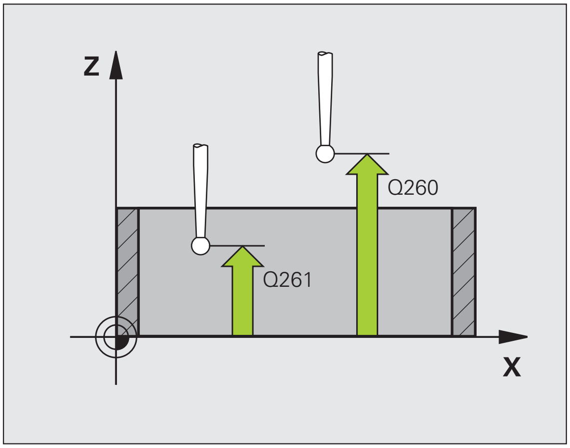
Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

The TNC always measures the first line in the direction of the minor axis of the working plane.

Cycle parameters





- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Spacing in 1st axis** Q326 (incremental): Distance between the first and second measuring points in the reference axis of the working plane. Input range 0 to 99999.9999
- **Spacing in 2nd axis** Q327 (incremental): Distance between third and fourth measuring points in the minor axis of the working plane. Input range 0 to 99999.9999
- **Corner** Q308: Number identifying the corner which the TNC is to set as datum. Input range 1 to 4
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **PREDEF**

- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
  Alternative: **PREDEF**
- **Execute basic rotation** Q304: Definition of whether the TNC should compensate workpiece misalignment with a basic rotation:
  0: No basic rotation
  1: Basic rotation
- **Number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the corner. If you enter Q305=0 and Q303=1, the TNC automatically sets the display so that the new datum is on the corner. If you enter Q305=0 and Q303=0, the TNC writes the corner to line 0 of the datum table. Input range 0 to 99999
- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the corner. Default setting = 0. Input range -99999.9999 to 99999.9999
- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the calculated corner. Default setting = 0. Input range -99999.9999 to 99999.9999
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  –1: Do not use. Is entered by the TNC when old programs are read in (see "Saving the calculated datum" on page 362).
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).

- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set datum in the touch probe axis
  1: Set the datum in the touch probe axis
- **Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999

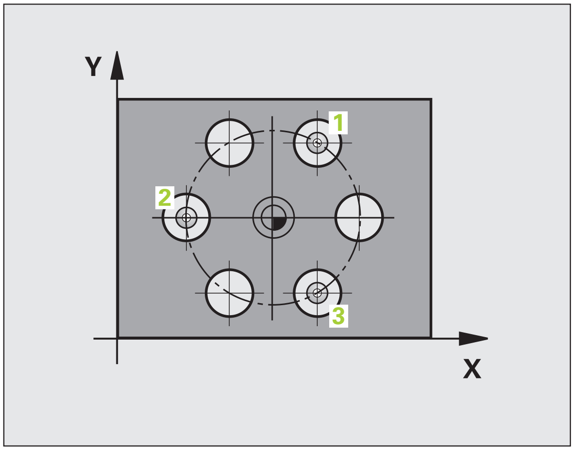
Example: NC blocks

```
5 TCH PROBE 415 DATUM INSIDE CORNER
    Q263=+37   ;1ST POINT 1ST AXIS
    Q264=+7    ;1ST POINT 2ND AXIS
    Q326=50    ;SPACING IN 1ST AXIS
    Q296=+95   ;3RD POINT 1ST AXIS
    Q297=+25   ;3RD POINT 2ND AXIS
    Q327=45    ;SPACING IN 2ND AXIS
    Q261=-5    ;MEASURING HEIGHT
    Q320=0     ;SET-UP CLEARANCE
    Q260=+20   ;CLEARANCE HEIGHT
    Q301=0     ;MOVE TO CLEARANCE
    Q304=0     ;BASIC ROTATION
    Q305=7     ;NO. IN TABLE
    Q331=+0    ;DATUM
    Q332=+0    ;DATUM
    Q303=+1    ;MEAS. VALUE TRANSFER
    Q381=1     ;PROBE IN TS AXIS
    Q382=+85   ;1ST CO. FOR TS AXIS
    Q383=+50   ;2ND CO. FOR TS AXIS
    Q384=+0    ;3RD CO. FOR TS AXIS
    Q333=+1    ;DATUM
```

15.10 DATUM CIRCLE CENTER (Cycle 416, DIN/ISO: G416)

Cycle run

Touch Probe Cycle 416 finds the center of a bolt hole circle by measuring three holes, and defines the determined center as the datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.



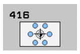
1 Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe at rapid traverse (value from MP6150) to the point entered as center of the first hole **1**
2 Then the touch probe moves to the entered measuring height and probes four points to find the first hole center.
3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
4 The TNC moves the touch probe to the entered measuring height and probes four points to find the second hole center.
5 The touch probe returns to the clearance height and then to the position entered as center of the third hole **3**.
6 The TNC moves the touch probe to the entered measuring height and probes four points to find the third hole center.
7 Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Saving the calculated datum" on page 362) and saves the actual values in the Q parameters listed below.
8 If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of bolt hole circle diameter

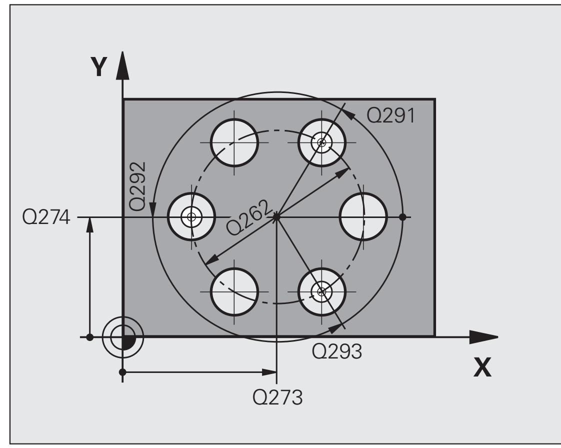
Please note while programming:

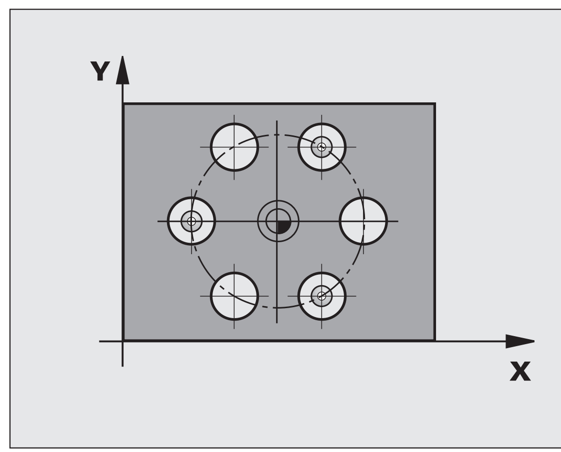
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **`Center in 1st axis`** Q273 (absolute): Bolt hole circle center (nominal value) in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **`Center in 2nd axis`** Q274 (absolute): Bolt hole circle center (nominal value) in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **`Nominal diameter`** Q262: Enter the approximate bolt-hole circle diameter. The smaller the hole diameter, the more exact the nominal diameter must be. Input range 0 to 99999.9999
- **`Angle of 1st hole`** Q291 (absolute): Polar coordinate angle of the first hole center in the working plane. Input range -360.0000 to 360.0000
- **`Angle of 2nd hole`** Q292 (absolute): Polar coordinate angle of the second hole center in the working plane. Input range -360.0000 to 360.0000
- **`Angle of 3rd hole`** Q293 (absolute): Polar coordinate angle of the third hole center in the working plane. Input range -360.0000 to 360.0000
- **`Measuring height in the touch probe axis`** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **`Clearance height`** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **`PREDEF`**
- **`Number in table`** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the bolt-hole circle center. If you enter Q305=0 and Q303=1, the TNC automatically sets the display so that the new datum is on the bolt hole center. If you enter Q305=0 and Q303=0, the TNC writes the bolt hole center to line 0 of the datum table. Input range 0 to 99999
- **`New datum for reference axis`** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the bolt-hole center. Default setting = 0. Input range -99999.9999 to 99999.9999





- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the bolt-hole center. Default setting = 0. Input range -99999.9999 to 99999.9999
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  –1: Do not use. Is entered by the TNC when old programs are read in (see "Saving the calculated datum" on page 362).
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).
- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set datum in the touch probe axis
  1: Set the datum in the touch probe axis
- **Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999

Example: NC blocks

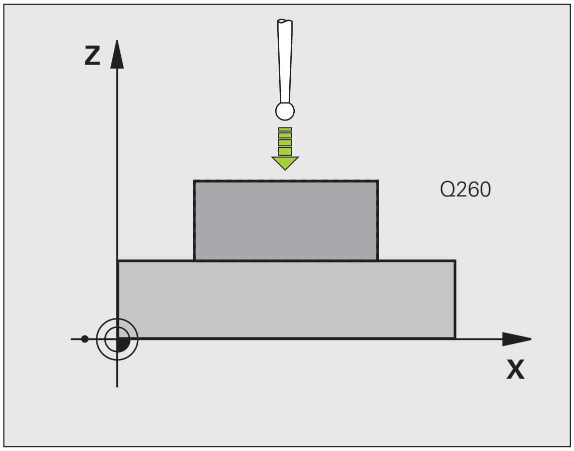
```
5 TCH PROBE 416 DATUM CIRCLE CENTER
     Q273=+50   ;CENTER IN 1ST AXIS
     Q274=+50   ;CENTER IN 2ND AXIS
     Q262=90    ;NOMINAL DIAMETER
     Q291=+34   ;ANGLE OF 1ST HOLE
     Q292=+70   ;ANGLE OF 2ND HOLE
     Q293=+210  ;ANGLE OF 3RD HOLE
     Q261=-5    ;MEASURING HEIGHT
     Q260=+20   ;CLEARANCE HEIGHT
     Q305=12    ;NO. IN TABLE
     Q331=+0    ;DATUM
     Q332=+0    ;DATUM
     Q303=+1    ;MEAS. VALUE TRANSFER
     Q381=1     ;PROBE IN TS AXIS
     Q382=+85   ;1ST CO. FOR TS AXIS
     Q383=+50   ;2ND CO. FOR TS AXIS
     Q384=+0    ;3RD CO. FOR TS AXIS
     Q333=+1    ;DATUM
     Q320=0     ;SET-UP CLEARANCE
```

- **Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140, and is only effective when the datum is probed in the touch probe axis. Input range 0 to 99999.9999; alternatively **PREDEF**

15.11 DATUM IN TOUCH PROBE AXIS (Cycle 417, DIN/ISO: G417)

Cycle run

Touch Probe Cycle 417 measures any coordinate in the touch probe axis and defines it as datum. If desired, the TNC can also enter the measured coordinate in a datum table or preset table.



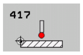
1. Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe to the programmed probe starting point **1** at rapid traverse (value from MP6150). The TNC offsets the touch probe by the safety clearance in the positive direction of the touch probe axis.
2. Then the touch probe moves in its own axis to the coordinate entered as touch point **1** and measures the actual position with a simple probing movement.
3. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Saving the calculated datum" on page 362) and saves the actual value in the Q parameter listed below.

Parameter number	Meaning
Q160	Actual value of measured point

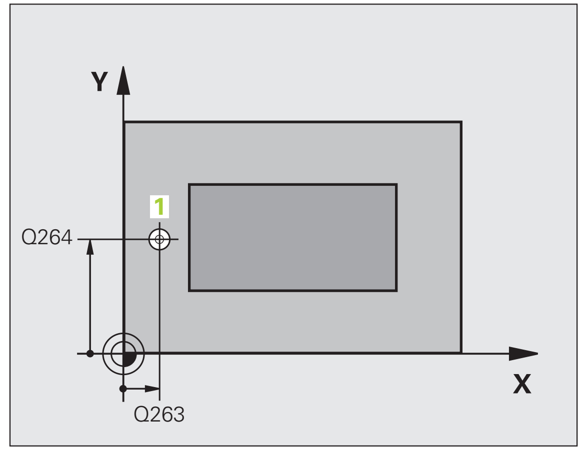
Please note while programming:

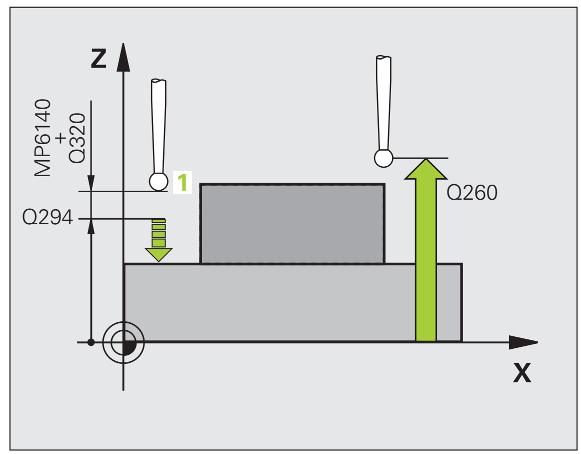
Before a cycle definition you must have programmed a tool call to define the touch probe axis. The TNC then sets the datum in this axis.

Cycle parameters



- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 3rd axis** Q294 (absolute): Coordinate of the first touch point in the touch probe axis. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **PREDEF**
- **Number in table** Q305: Enter the number in the datum or preset table in which the TNC is to save the coordinate. If you enter Q305=0 and Q303=1, the TNC automatically sets the display so that the new datum is on the probed surface. If you enter Q305=0 and Q303=0, the TNC writes the coordinates to line 0 of the datum table. Input range 0 to 99999
- **New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  –1: Do not use. Is entered by the TNC when old programs are read in (see "Saving the calculated datum" on page 362).
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).





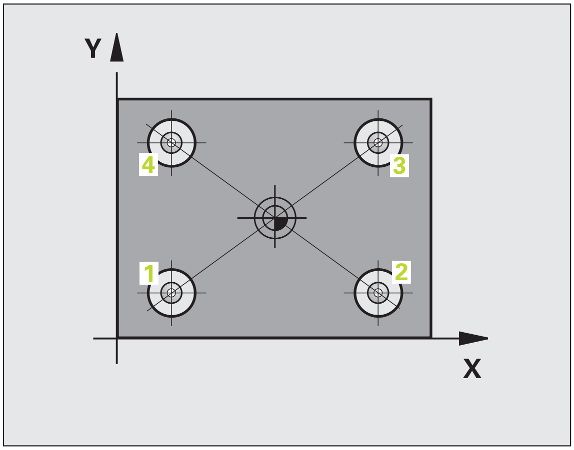
Example: NC blocks

```
5 TCH PROBE 417 DATUM IN TS AXIS
     Q263=+25   ;1ST POINT 1ST AXIS
     Q264=+25   ;1ST POINT 2ND AXIS
     Q294=+25   ;1ST POINT 3RD AXIS
     Q320=0     ;SET-UP CLEARANCE
     Q260=+50   ;CLEARANCE HEIGHT
     Q305=0     ;NO. IN TABLE
     Q333=+0    ;DATUM
     Q303=+1    ;MEAS. VALUE TRANSFER
```

15.12 DATUM AT CENTER OF 4 HOLES (Cycle 418, DIN/ISO: G418)

Cycle run

Touch Probe Cycle 418 calculates the intersection of the lines connecting opposite holes and sets the datum at the intersection. If desired, the TNC can also enter the intersection into a datum table or preset table.



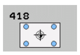
1. Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe at rapid traverse (value from MP6150) to the center of the first hole **1**.
2. Then the touch probe moves to the entered measuring height and probes four points to find the first hole center.
3. The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
4. The TNC moves the touch probe to the entered measuring height and probes four points to find the second hole center.
5. The TNC repeats steps 3 and 4 for the holes **3** and **4**.
6. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Saving the calculated datum" on page 362). The TNC calculates the datum as the intersection of the lines connecting the centers of holes **1**/**3** and **2**/**4** and saves the actual values in the Q parameters listed below.
7. If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

Parameter number	Meaning
Q151	Actual value of intersection point in reference axis
Q152	Actual value of intersection point in minor axis

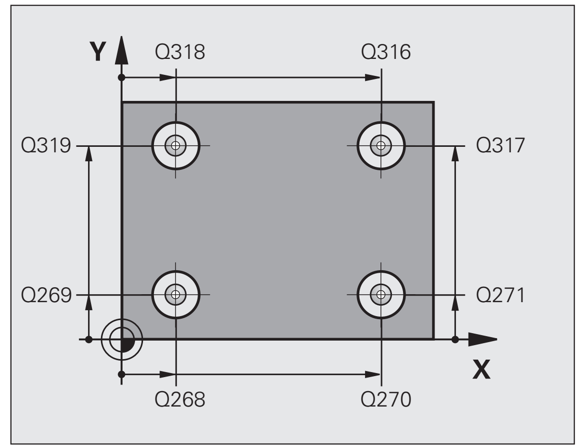
Please note while programming:

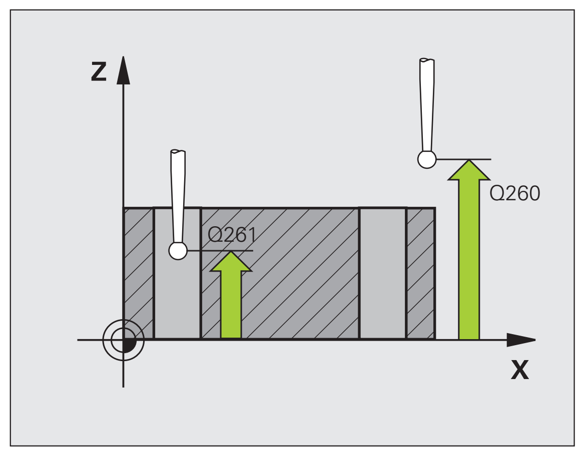
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **First center in 1st axis** Q268 (absolute): Center of the 1st hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **First center in 2nd axis** Q269 (absolute): Center of the 1st hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Second center in 1st axis** Q270 (absolute): Center of the 2nd hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Second center in 2nd axis** Q271 (absolute): Center of the 2nd hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **3rd center in 1st axis** Q316 (absolute): center of the 3rd hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Third center in 2nd axis** Q317 (absolute): Center of the 3rd hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Fourth center in 1st axis** Q318 (absolute): Center of the 4th hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Fourth center in 2nd axis** Q319 (absolute): Center of the 4th hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **PREDEF**





- **Number in table** Q305: Enter the number in the datum or preset table in which the TNC is to save the coordinates of the line intersection. If you enter Q305=0 and Q303=1, the TNC automatically sets the display so that the new datum is at the intersection of the connecting lines. If you enter Q305=0 and Q303=0, the TNC writes the coordinates of the intersection of the connecting lines to line 0 of the datum table. Input range 0 to 99999
- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the calculated intersection of the connecting lines. Default setting = 0. Input range -99999.9999 to 99999.9999
- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the calculated intersection of the connecting lines. Default setting = 0. Input range -99999.9999 to 99999.9999
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  –1: Do not use. Is entered by the TNC when old programs are read in (see "Saving the calculated datum" on page 362).
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).

- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set datum in the touch probe axis
  1: Set the datum in the touch probe axis
- **Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1
- **Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **Probe TS axis: Coord. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1. Input range -99999.9999 to 99999.9999
- **New datum in TS axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999

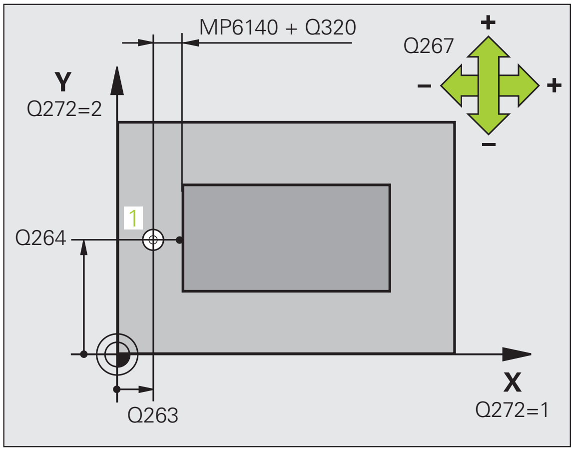
Example: NC blocks

```
5 TCH PROBE 418 DATUM FROM 4 HOLES
    Q268=+20  ;1ST CENTER IN 1ST AXIS
    Q269=+25  ;1ST CENTER IN 2ND AXIS
    Q270=+150 ;2ND CENTER IN 1ST AXIS
    Q271=+25  ;2ND CENTER IN 2ND AXIS
    Q316=+150 ;3RD CENTER IN 1ST AXIS
    Q317=+85  ;3RD CENTER IN 2ND AXIS
    Q318=+22  ;4TH CENTER IN 1ST AXIS
    Q319=+80  ;4TH CENTER IN 2ND AXIS
    Q261=-5   ;MEASURING HEIGHT
    Q260=+10  ;CLEARANCE HEIGHT
    Q305=12   ;NO. IN TABLE
    Q331=+0   ;DATUM
    Q332=+0   ;DATUM
    Q303=+1   ;MEAS. VALUE TRANSFER
    Q381=1    ;PROBE IN TS AXIS
    Q382=+85  ;1ST CO. FOR TS AXIS
    Q383=+50  ;2ND CO. FOR TS AXIS
    Q384=+0   ;3RD CO. FOR TS AXIS
    Q333=+0   ;DATUM
```

15.13 DATUM IN ONE AXIS (Cycle 419, DIN/ISO: G419)

Cycle run

Touch Probe Cycle 419 measures any coordinate in any axis and defines it as datum. If desired, the TNC can also enter the measured coordinate in a datum table or preset table.



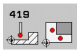
1. Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe to the programmed probe starting point **1** at rapid traverse (value from MP6150). The TNC offsets the touch probe by the safety clearance in the direction opposite the programmed probing direction.
2. Then the touch probe moves to the programmed measuring height and measures the actual position with a simple probing movement.
3. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Saving the calculated datum" on page 362).

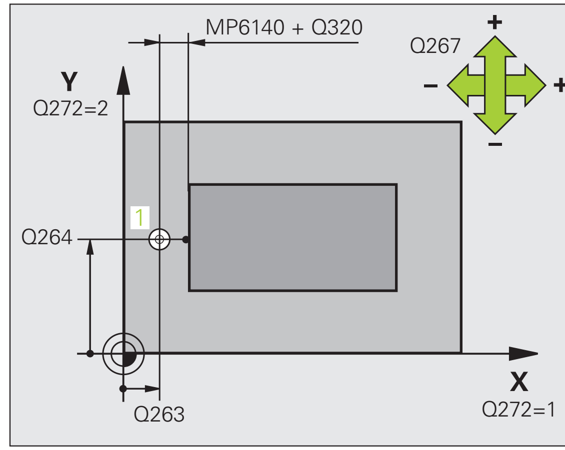
Please note while programming:

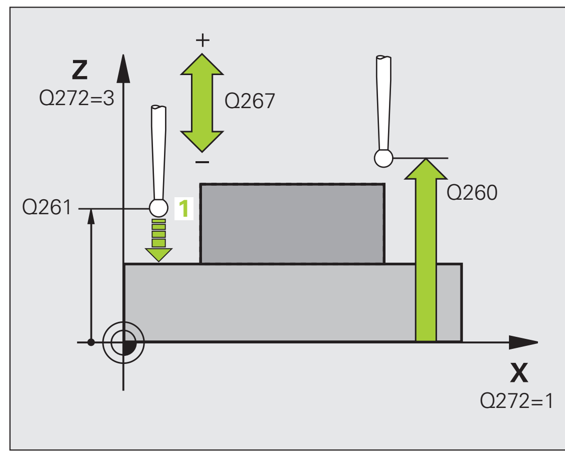
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

If you use Cycle 419 several times in succession to save the datum in more than one axis in the preset table, you must activate the preset number last written to by Cycle 419 after every execution of Cycle 419 (this is not required if you overwrite the active preset).

Cycle parameters







- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **PREDEF**
- **Measuring axis (1 to 3: 1=reference axis)** Q272: Axis in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
  3: Touch probe axis = measuring axis

Axis assignment

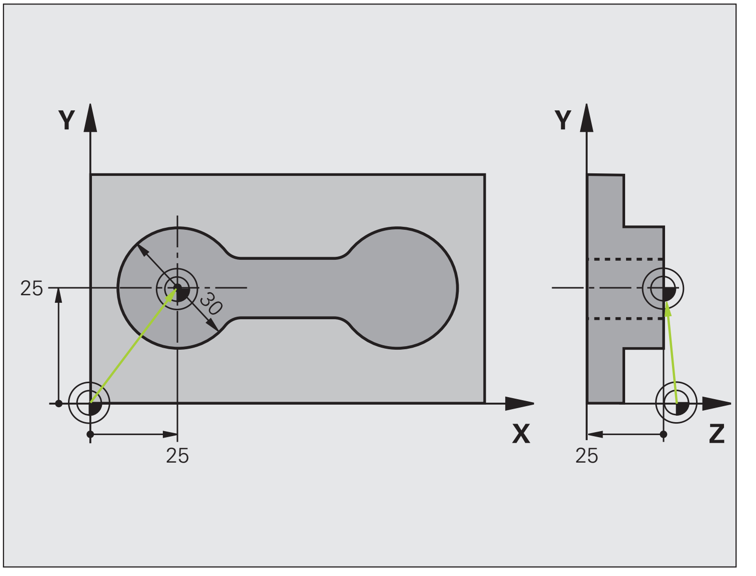
Active touch probe axis: Q272= 3	Corresponding reference axis: Q272 = 1	Corresponding minor axis: Q272 = 2
Z	X	Y
Y	Z	X
X	Y	Z

- **Traverse direction** Q267: Direction in which the touch probe is to approach the workpiece:
  –1: Negative traverse direction
  +1: Positive traverse direction
- **Number in table** Q305: Enter the number in the datum or preset table in which the TNC is to save the coordinate. If you enter Q305=0 and Q303=1, the TNC automatically sets the display so that the new datum is on the probed surface. If you enter Q305=0 and Q303=0, the TNC writes the coordinates to line 0 of the datum table. Input range 0 to 99999
- **New datum** Q333 (absolute): Coordinate at which the TNC should set the datum. Default setting = 0. Input range -99999.9999 to 99999.9999
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  –1: Do not use. See "Saving the calculated datum" on page 362
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).

Example: NC blocks

```
5 TCH PROBE 419 DATUM IN ONE AXIS
    Q263=+25  ;1ST POINT 1ST AXIS
    Q264=+25  ;1ST POINT 2ND AXIS
    Q261=+25  ;MEASURING HEIGHT
    Q320=0    ;SET-UP CLEARANCE
    Q260=+50  ;CLEARANCE HEIGHT
    Q272=+1   ;MEASURING AXIS
    Q267=+1   ;TRAVERSE DIRECTION
    Q305=0    ;NO. IN TABLE
    Q333=+0   ;DATUM
    Q303=+1   ;MEAS. VALUE TRANSFER
```

Example: Datum setting in center of a circular segment and on top surface of workpiece

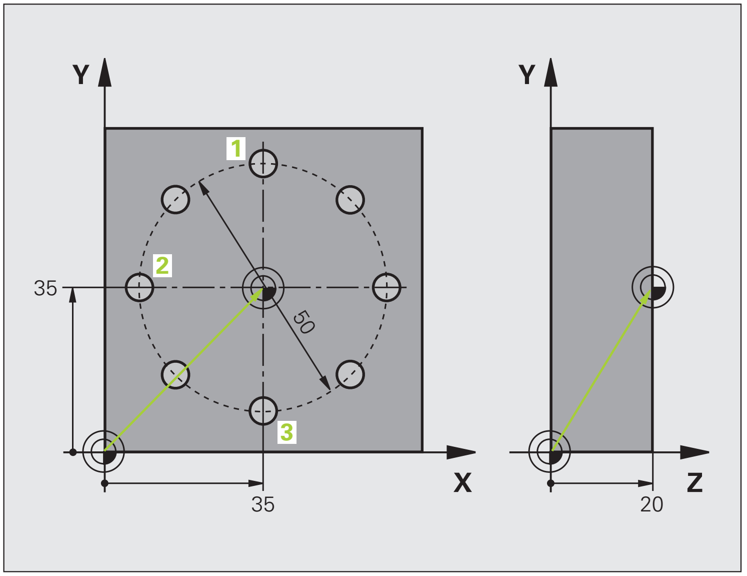


0 BEGIN PGM CYC413 MM	
1 TOOL CALL 69 Z	Call tool 0 to define the touch probe axis

2 TCH PROBE 413 DATUM OUTSIDE CIRCLE	
Q321=+25 ;CENTER IN 1ST AXIS	Center of circle: X coordinate
Q322=+25 ;CENTER IN 2ND AXIS	Center of circle: Y coordinate
Q262=30 ;NOMINAL DIAMETER	Circle diameter
Q325=+90 ;STARTING ANGLE	Polar coordinate angle for 1st touch point
Q247=+45 ;STEPPING ANGLE	Stepping angle for calculating the touch points 2 to 4
Q261=-5 ;MEASURING HEIGHT	Coordinate in the touch probe axis in which the measurement is made
Q320=2 ;SET-UP CLEARANCE	Safety clearance in addition to MP6140
Q260=+10 ;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision
Q301=0 ;MOVE TO CLEARANCE	Do not move to clearance height between measuring points
Q305=0 ;NO. IN TABLE	Set display
Q331=+0 ;DATUM	Set the display in X to 0
Q332=+10 ;DATUM	Set the display in Y to 10
Q303=+0 ;MEAS. VALUE TRANSFER	Without function, since display is to be set
Q381=1 ;PROBE IN TS AXIS	Also set datum in the touch probe axis
Q382=+25 ;1ST CO. FOR TS AXIS	X coordinate of touch point
Q383=+25 ;2ND CO. FOR TS AXIS	Y coordinate of touch point
Q384=+25 ;3RD CO. FOR TS AXIS	Z coordinate of touch point
Q333=+0 ;DATUM	Set the display in Z to 0
Q423=4 ;NO. OF MEAS. POINTS	Number of measuring points
Q365=1 ;TYPE OF TRAVERSE	Position in circular arc or in straight line to the next touch point
3 CALL PGM 35K47	Call part program
4 END PGM CYC413 MM	

Example: Datum setting on top surface of workpiece and in center of a bolt hole circle

The measured bolt-hole circle center shall be written in the preset table so that it may be used at a later time.



0 BEGIN PGM CYC416 MM	
1 TOOL CALL 69 Z	Call tool 0 to define the touch probe axis
2 TCH PROBE 417 DATUM IN TS AXIS	Cycle definition for datum setting in the touch probe axis
Q263=+7.5 ;1ST POINT 1ST AXIS	Touch point: X coordinate
Q264=+7.5 ;1ST POINT 2ND AXIS	Touch point: Y coordinate
Q294=+25 ;1ST POINT 3RD AXIS	Touch point: Z coordinate
Q320=0 ;SET-UP CLEARANCE	Safety clearance in addition to MP6140
Q260=+50 ;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision
Q305=1 ;NO. IN TABLE	Write Z coordinate in line 1
Q333=+0 ;DATUM	Set touch-probe axis to 0
Q303=+1 ;MEAS. VALUE TRANSFER	In the preset table PRESET.PR, save the calculated datum referenced to the machine-based coordinate system (REF system)

3 TCH PROBE 416 DATUM CIRCLE CENTER	
Q273=+35 ;CENTER IN 1ST AXIS	Center of the bolt hole circle: X coordinate
Q274=+35 ;CENTER IN 2ND AXIS	Center of the bolt hole circle: Y coordinate
Q262=50 ;NOMINAL DIAMETER	Diameter of the bolt hole circle
Q291=+90 ;ANGLE OF 1ST HOLE	Polar coordinate angle for 1st hole center **1**
Q292=+180 ;ANGLE OF 2ND HOLE	Polar coordinate angle for 2nd hole center **2**
Q293=+270 ;ANGLE OF 3RD HOLE	Polar coordinate angle for 3rd hole center **3**
Q261=+15 ;MEASURING HEIGHT	Coordinate in the touch probe axis in which the measurement is made
Q260=+10 ;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision
Q305=1 ;NO. IN TABLE	Enter center of bolt hole circle (X and Y) in line 1
Q331=+0 ;DATUM	
Q332=+0 ;DATUM	
Q303=+1 ;MEAS. VALUE TRANSFER	In the preset table PRESET.PR, save the calculated datum referenced to the machine-based coordinate system (REF system)
Q381=0 ;PROBE IN TS AXIS	Do not set a datum in the touch probe axis
Q382=+0 ;1ST CO. FOR TS AXIS	No function
Q383=+0 ;2ND CO. FOR TS AXIS	No function
Q384=+0 ;3RD CO. FOR TS AXIS	No function
Q333=+0 ;DATUM	No function
Q320=0 ;SET-UP CLEARANCE	Safety clearance in addition to MP6140
4 CYCL DEF 247 DATUM SETTING	Activate new preset with Cycle 247
Q339=1 ;DATUM NUMBER	
6 CALL PGM 35KLZ	Call part program
7 END PGM CYC416 MM	

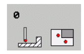
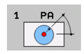
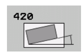
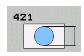
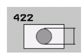
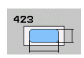
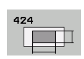
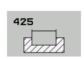
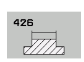
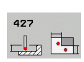
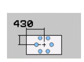
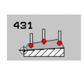
16

Touch Probe Cycles: Automatic Workpiece Inspection

16.1 Fundamentals

Overview

The TNC offers twelve cycles for measuring workpieces automatically.

Cycle	Soft key	Page
0 REFERENCE PLANE Measuring a coordinate in a selectable axis		Page 420
1 POLAR DATUM PLANE Measuring a point in a probing direction		Page 421
420 MEASURE ANGLE Measuring an angle in the working plane		Page 423
421 MEASURE HOLE Measuring the position and diameter of a hole		Page 426
422 MEASURE CIRCLE OUTSIDE Measuring the position and diameter of a circular stud		Page 430
423 MEASURE RECTANGLE INSIDE Measuring the position, length and width of a rectangular pocket		Page 434
424 MEASURE RECTANGLE OUTSIDE Measuring the position, length and width of a rectangular stud		Page 438
425 MEASURE INSIDE WIDTH (2nd soft-key row) Measuring slot width		Page 442
426 MEASURE RIDGE WIDTH (2nd soft-key row) Measuring the width of a ridge		Page 445
427 MEASURE COORDINATE (2nd soft-key row) Measuring any coordinate in a selectable axis		Page 448
430 MEASURE BOLT HOLE CIRCLE (2nd soft-key row) Measuring position and diameter of a bolt hole circle		Page 451
431 MEASURE PLANE (2nd soft-key row) Measuring the A and B axis angles of a plane		Page 455

Recording the results of measurement

For all cycles in which you automatically measure workpieces (with the exception of Cycles 0 and 1), you can have the TNC record the measurement results. In the respective probing cycle you can define if the TNC is to

- Save the measuring log to a file
- Interrupt program run and display the measuring log on the screen
- Create no measuring log

If you want to save the measuring log as a file, the TNC, by default, saves the measuring log as an ASCII file in the directory from which you run the measuring program. As an alternative, you can also send the measuring log directly to a printer or transfer it to a PC via the data interface. To do this, set the print function (in the interface configuration menu) to RS232:\ (see also the User's Manual under “MOD Functions, Setting Up the Data Interface”).

All measured values listed in the log file are referenced to the datum active during the respective cycle you are running. In addition, the coordinate system may have been rotated in the plane or the plane may have been tilted by using 3-D ROT. In this case, the TNC converts the measuring results to the respective active coordinate system.

Use the HEIDENHAIN data transfer software TNCremo if you wish to output the measuring log via the data interface.

Example: Measuring log for touch probe cycle 421:

Measuring log for Probing Cycle 421 Hole Measuring

Date: 30-06-2005
Time: 6:55:04
Measuring program: TNC:\GEH35712\CHECK1.H

Nominal values:
Center in reference axis: 50.0000
Center in minor axis: 65.0000
Diameter: 12.0000

Given limit values:
Maximum dimension for center in reference axis: 50.1000
Min. limit for center in reference axis: 49.9000
Maximum dimension for center in minor axis: 65.1000
Min. limit for center in minor axis: 64.9000
Maximum dimension for hole: 12.0450
Minimum dimension for hole: 12.0000

Actual values:Center
Reference axis: 50.0810
Center in minor axis: 64.9530
Diameter: 12.0259

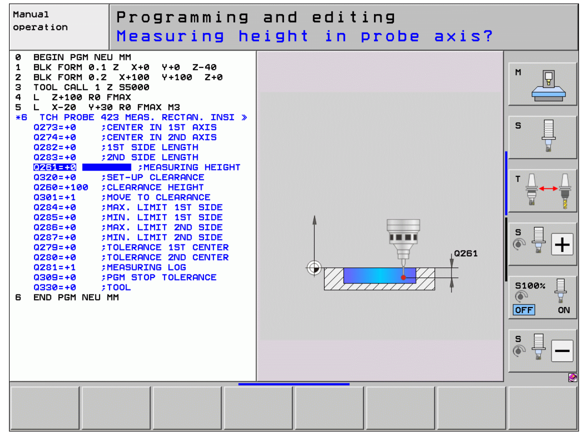
Deviations:
Center in reference axis: 0.0810
Center in minor axis: -0.0470
Diameter: 0.0259

Further measuring results: Measuring height: -5.0000

End of measuring log

Measurement results in Q parameters

The TNC saves the measurement results of the respective touch probe cycle in the globally effective Q parameters Q150 to Q160. Deviations from the nominal value are saved in the parameters Q161 to Q166. Note the table of result parameters listed with every cycle description.



During cycle definition the TNC also shows the result parameters for the respective cycle in a help graphic (see figure at upper right). The highlighted result parameter belongs to that input parameter.

Classification of results

For some cycles you can inquire the status of measuring results through the globally effective Q parameters Q180 to Q182.:

Class of results	Parameter value
Measurement results are within tolerance	Q180 = 1
Rework is required	Q181 = 1
Scrap	Q182 = 1

The TNC sets the rework or scrap marker as soon as one of the measuring values falls outside of tolerance. To determine which of the measuring results lies outside of tolerance, check the measuring log, or compare the respective measuring results (Q150 to Q160) with their limit values.

In Cycle 427 the TNC assumes that you are measuring an outside dimension (stud). However, you can correct the status of the measurement by entering the correct maximum and minimum dimension together with the probing direction.

The TNC also sets the status markers if you have not defined any tolerance values or maximum/minimum dimensions.

Tolerance monitoring

For most of the cycles for workpiece inspection you can have the TNC perform tolerance monitoring. This requires that you define the necessary limit values during cycle definition. If you do not wish to monitor for tolerances, simply leave the 0 (the default value) in the monitoring parameters.

Tool monitoring

For some cycles for workpiece inspection you can have the TNC perform tool monitoring. The TNC then monitors whether

- The tool radius should be compensated because of the deviations from the nominal value (values in Q16x).
- The deviations from the nominal value (values in Q16x) are greater than the tool breakage tolerance.

Tool compensation

This function works only:

- If the tool table is active.
- If tool monitoring is switched on in the cycle (enter a tool name or **Q330** unequal to 0). Select the tool name input by soft key. The TNC no longer displays the right single quotation mark.

If you perform several compensation measurements, the TNC adds the respective measured deviation to the value stored in the tool table.

The TNC always compensates the tool radius in the DR column of the tool table, even if the measured deviation lies within the given tolerance. You can inquire whether re-working is necessary via parameter Q181 in the NC program (Q181=1: must be reworked).

For Cycle 427:

- If an axis of the active working plane is defined as measuring axis (Q272 = 1 or 2), the TNC compensates the tool radius as described above. From the defined traversing direction (Q267) the TNC determines the direction of compensation.
- If the touch probe axis is defined as measuring axis (Q272 = 3), the TNC compensates the tool length.

Tool breakage monitoring

> This function works only:
>
> - If the tool table is active.
> - If tool monitoring is switched on in the cycle (enter Q330 not equal to 0).
> - If the breakage tolerance RBREAK for the tool number entered in the table is greater than 0 (see also the User's Manual, section 5.2 "Tool Data").

The TNC will output an error message and stop program run if the measured deviation is greater than the breakage tolerance of the tool. At the same time the tool will be deactivated in the tool table (column TL = L).

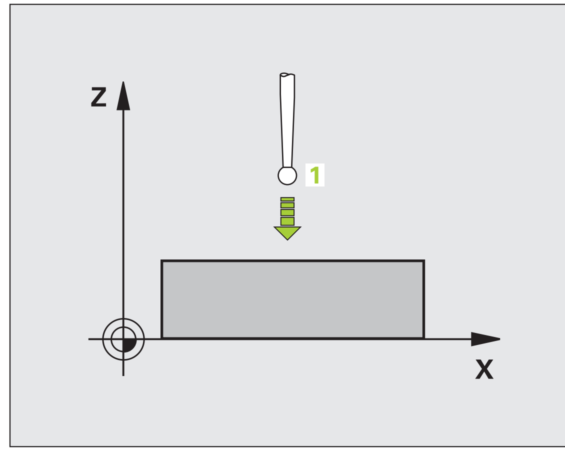
Reference system for measurement results

The TNC transfers all the measurement results to the result parameters and the log file in the active coordinate system, or as the case may be, the shifted and/or rotated/tilted coordinate system.

16.2 REF. PLANE (Cycle 0, DIN/ISO: G55)

Cycle run

1. The touch probe moves at rapid traverse (value from MP6150) to the starting position **1** programmed in the cycle.
2. Then the touch probe approaches the workpiece at the feed rate assigned in MP6120. The probing direction is defined in the cycle.
3. After the TNC has saved the position, the touch probe retracts to the starting point and saves the measured coordinate in a Q parameter. The TNC also stores the coordinates of the touch probe position at the time of the triggering signal in the parameters Q115 to Q119. For the values in these parameters the TNC does not account for the stylus length and radius.

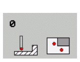


Please note while programming:

Danger of collision!

Pre-position the touch probe in order to avoid a collision when the programmed pre-positioning point is approached.

Cycle parameters



- **`Parameter number for result`**: Enter the number of the Q parameter to which you want to assign the coordinate. Input range: 0 to 1999
- **`Probing axis/Probing direction`**: Enter the probing axis with the axis selection keys or ASCII keyboard and the algebraic sign for the probing direction. Confirm your entry with the ENT key. Input range: All NC axes
- **`Nominal position value`**: Use the axis selection keys or the ASCII keyboard to enter all coordinates of the nominal pre-positioning point values for the touch probe. Input range -99999.9999 to 99999.9999
- To conclude the input, press the ENT key.

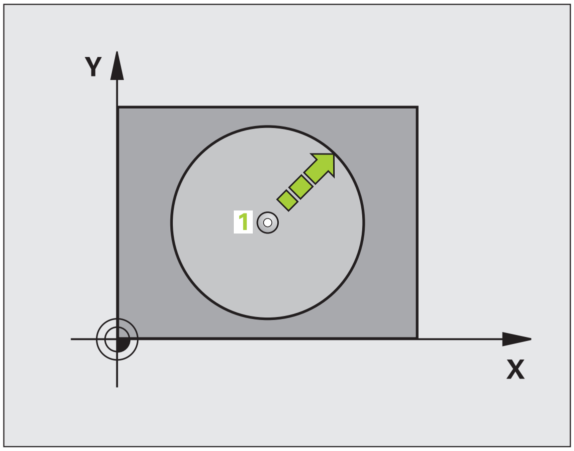
Example: NC blocks

```
67 TCH PROBE 0.0 REF. PLANE Q5 X-
68 TCH PROBE 0.1 X+5 Y+0 Z-5
```

16.3 POLAR REFERENCE PLANE (Cycle 1)

Cycle run

Touch Probe Cycle 1 measures any position on the workpiece in any direction.



1 The touch probe moves at rapid traverse (value from MP6150) to the starting position **1** programmed in the cycle.
2 Then the touch probe approaches the workpiece at the feed rate assigned in MP6120. During probing the TNC moves simultaneously in two axes (depending on the probing angle). The probing direction is defined by the polar angle entered in the cycle.
3 After the TNC has saved the position, the probe returns to the starting point. The TNC also stores the coordinates of the touch probe position at the time of the triggering signal in parameters Q115 to Q119.

Please note while programming:

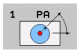
Danger of collision!

Pre-position the touch probe in order to avoid a collision when the programmed pre-positioning point is approached.

The probing axis defined in the cycle specifies the probing plane:

- Probing axis X: X/Y plane
- Probing axis Y: Y/Z plane
- Probing axis Z: Z/X plane

Cycle parameters



- **`Probing axis`**: Enter the probing axis with the axis selection keys or ASCII keyboard. Confirm your entry with the ENT key. Input range: **`X`**, **`Y`** or **`Z`**
- **`Probing angle`**: Angle, measured from the probing axis, at which the touch probe is to move. Input range –180.0000 to 180.0000
- **`Nominal position value`**: Use the axis selection keys or the ASCII keyboard to enter all coordinates of the nominal pre-positioning point values for the touch probe. Input range -99999.9999 to 99999.9999
- To conclude the input, press the ENT key.

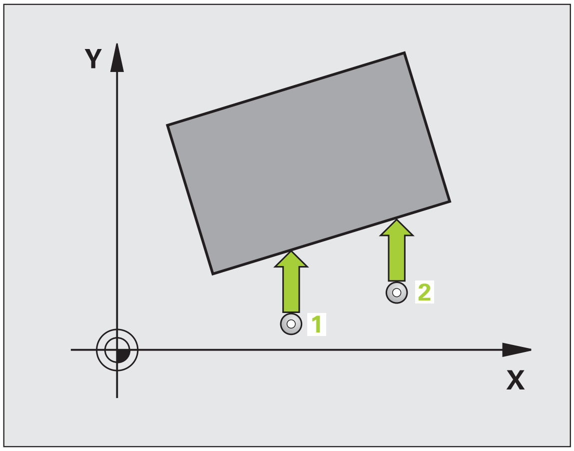
Example: NC blocks

```
67 TCH PROBE 1.0 POLAR REFERENCE PLANE
68 TCH PROBE 1.1 X ANGLE: +30
69 TCH PROBE 1.2 X+5 Y+0 Z-5
```

16.4 MEASURE ANGLE (Cycle 420, DIN/ISO: G420)

Cycle run

Touch Probe Cycle 420 measures the angle that any straight surface on the workpiece describes with respect to the reference axis of the working plane.



1. Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe to the programmed probe starting point **1** at rapid traverse (value from MP6150). The TNC offsets the touch probe by the safety clearance in the direction opposite to the defined traverse direction.
2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120).
3. Then the touch probe moves to the next starting position **2** and probes the second touch point.
4. The TNC returns the touch probe to the clearance height and saves the measured angle in the following Q parameter:

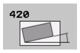
Parameter number	Meaning
Q150	The measured angle is referenced to the reference axis of the machining plane.

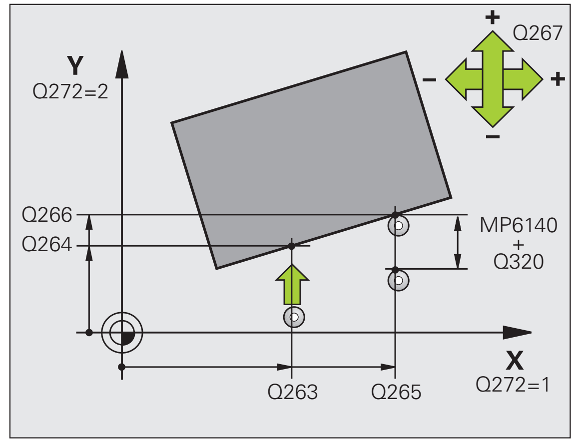
Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

If touch probe axis = measuring axis, set **Q263** equal to **Q265** if the angle about the A axis is to be measured; set **Q263** not equal to **Q265** if the angle is to be measured about the B axis.

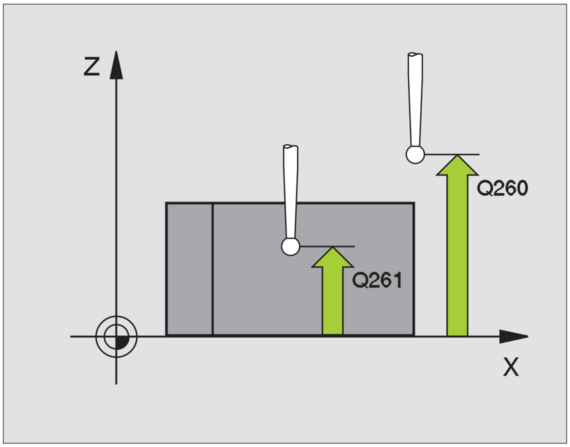
Cycle parameters





- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 1st axis** Q265 (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 2nd axis** Q266 (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Measuring axis** Q272: Axis in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
  3: Touch probe axis = measuring axis

- **Traverse direction 1** Q267: Direction in which the touch probe is to approach the workpiece:
  –1: Negative traverse direction
  +1: Positive traverse direction
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **PREDEF**
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
  Alternative: **PREDEF**
- **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
  0: No measuring log
  1: Create measuring log: By default, the TNC saves the **log file TCHPR420.TXT** in the directory in which your measuring program is stored.
  2: Interrupt program run and display the measuring log on the screen. Resume program run with NC Start.



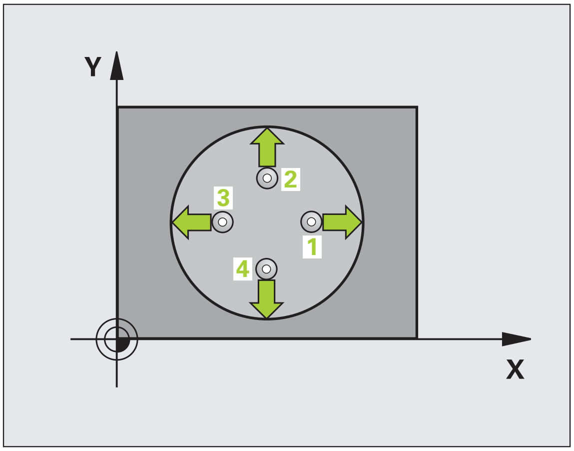
Example: NC blocks

```
5 TCH PROBE 420 MEASURE ANGLE
    Q263=+10  ;1ST POINT 1ST AXIS
    Q264=+10  ;1ST POINT 2ND AXIS
    Q265=+15  ;2ND POINT 1ST AXIS
    Q266=+95  ;2ND POINT 2ND AXIS
    Q272=1    ;MEASURING AXIS
    Q267=-1   ;TRAVERSE DIRECTION
    Q261=-5   ;MEASURING HEIGHT
    Q320=0    ;SET-UP CLEARANCE
    Q260=+10  ;CLEARANCE HEIGHT
    Q301=1    ;MOVE TO CLEARANCE
    Q281=1    ;MEASURING LOG
```

16.5 MEASURE HOLE (Cycle 421, DIN/ISO: G421)

Cycle run

Touch Probe Cycle 421 measures the center and diameter of a hole (or circular pocket). If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation values in system parameters.



1 Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe to the probe starting point **1** at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.
2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120). The TNC derives the probing direction automatically from the programmed starting angle.
3 Then the touch probe moves in a circular arc either at measuring height or at clearance height to the next starting point **2** and probes the second touch point.
4 The TNC positions the touch probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
5 Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviations in the following Q parameters:

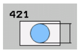
Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q163	Deviation from diameter

Please note while programming:

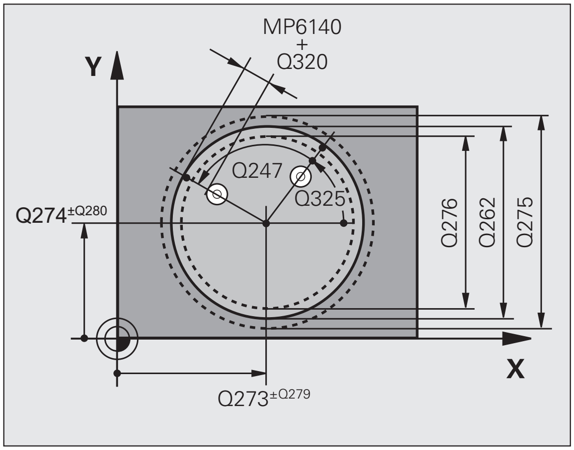
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

The smaller the angle, the less accurately the TNC can calculate the hole dimensions. Minimum input value: 5°.

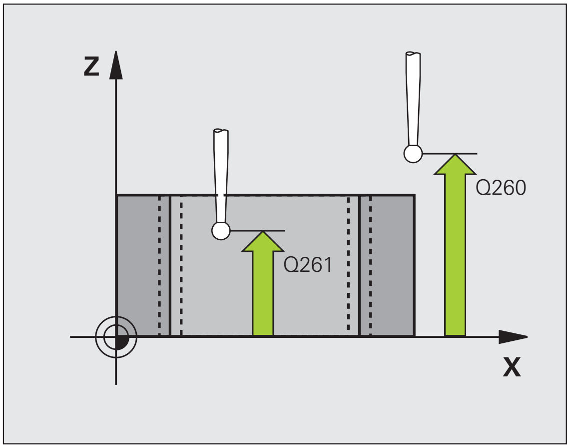
Cycle parameters



- **`Center in 1st axis`** Q273 (absolute): Center of the hole in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **`Center in 2nd axis`** Q274 (absolute value): Center of the hole in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **`Nominal diameter`** Q262: Enter the diameter of the hole. Input range 0 to 99999.9999
- **`Starting angle`** Q325 (absolute): Angle between the reference axis of the working plane and the first touch point. Input range -360.0000 to 360.0000
- **`Stepping angle`** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise). If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range -120.0000 to 120.0000



- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **PREDEF**
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
  Alternative: **PREDEF**
- **Maximum limit of size for hole** Q275: Maximum permissible diameter for the hole (circular pocket). Input range 0 to 99999.9999
- **Minimum limit of size for hole** Q276: Minimum permissible diameter for the hole (circular pocket). Input range 0 to 99999.9999
- **Tolerance for center 1st axis** Q279: Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999
- **Tolerance for center 2nd axis** Q280: Permissible position deviation in the minor axis of the working plane. Input range 0 to 99999.9999



- **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
  0: No measuring log
  1: Create measuring log: By default, the TNC saves the **log file TCHPR421.TXT** in the directory in which your measuring program is stored.
  2: Interrupt program run and display the measuring log on the screen. Resume program run with NC Start.
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
  0 Do not interrupt program run, no error message
  1: Interrupt program run, output an error message
- **Tool for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring" on page 418). Input range: 0 to 32767.9, alternatively tool name with max. 16 characters
  0: Monitoring not active
  >0: Tool number in the tool table TOOL.T
- **No. of measuring points (4/3)** Q423: Specify whether the TNC should measure the hole with 4 or 3 probing points:
  4: Use 4 measuring points (standard setting)
  3: Use 3 measuring points
- **Type of traverse? Line=0/Arc=1** Q365: Definition of the path function with which the touch probe is to move between the measuring points if "traverse to clearance height" (Q301=1) is active.
  0: Move in a straight line between measuring points
  1: Move in a circular arc on the pitch circle diameter between measuring points

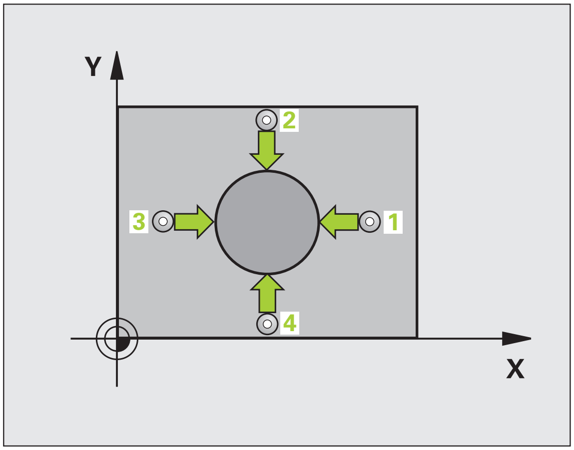
Example: NC blocks

```
5 TCH PROBE 421 MEASURE HOLE
    Q273=+50  ;CENTER IN 1ST AXIS
    Q274=+50  ;CENTER IN 2ND AXIS
    Q262=75   ;NOMINAL DIAMETER
    Q325=+0   ;STARTING ANGLE
    Q247=+60  ;STEPPING ANGLE
    Q261=-5   ;MEASURING HEIGHT
    Q320=0    ;SET-UP CLEARANCE
    Q260=+20  ;CLEARANCE HEIGHT
    Q301=1    ;MOVE TO CLEARANCE
    Q275=75.12;MAX. LIMIT
    Q276=74.95;MIN. LIMIT
    Q279=0.1  ;TOLERANCE 1ST CENTER
    Q280=0.1  ;TOLERANCE 2ND CENTER
    Q281=1    ;MEASURING LOG
    Q309=0    ;PGM STOP IF ERROR
    Q330=0    ;TOOL
    Q423=4    ;NO. OF MEAS. POINTS
    Q365=1    ;TYPE OF TRAVERSE
```

16.6 MEAS. CIRCLE OUTSIDE (Cycle 422, DIN/ISO: G422)

Cycle run

Touch Probe Cycle 422 measures the center and diameter of a circular stud. If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation values in system parameters.



1 Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe to the probe starting point **1** at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.
2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120). The TNC derives the probing direction automatically from the programmed starting angle.
3 Then the touch probe moves in a circular arc either at measuring height or at clearance height to the next starting point **2** and probes the second touch point.
4 The TNC positions the touch probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
5 Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviations in the following Q parameters:

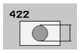
Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q163	Deviation from diameter

Please note while programming:

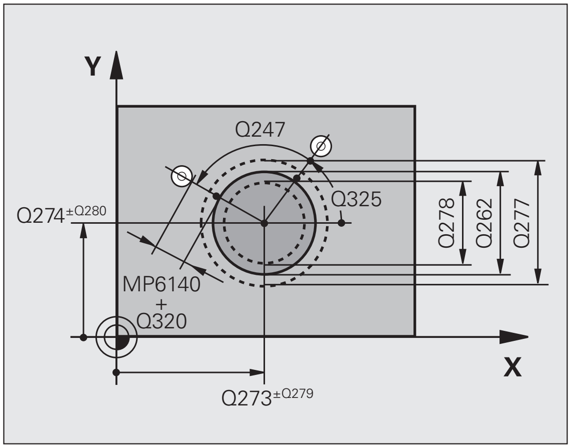
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

The smaller the angle, the less accurately the TNC can calculate the dimensions of the stud. Minimum input value: 5°

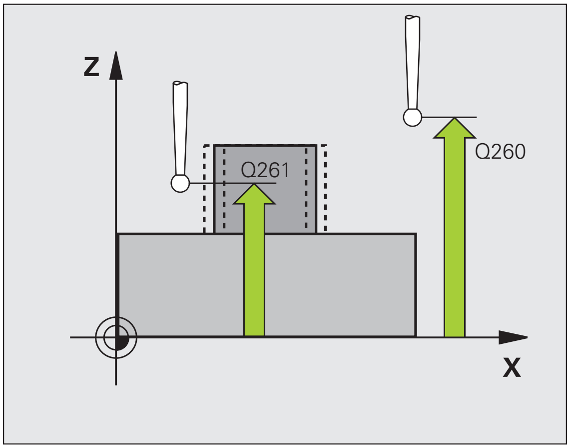
Cycle parameters



- **`Center in 1st axis`** Q273 (absolute): Center of the stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **`Center in 2nd axis`** Q274 (absolute): Center of the stud in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **`Nominal diameter`** Q262: Enter the diameter of the stud. Input range 0 to 99999.9999
- **`Starting angle`** Q325 (absolute): Angle between the reference axis of the working plane and the first touch point. Input range -360.0000 to 360.0000
- **`Stepping angle`** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise). If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range -120.0000 to 120.0000



- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **PREDEF**
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
  Alternative: **PREDEF**
- **Maximum limit of size for stud** Q277: Maximum permissible diameter for the stud. Input range 0 to 99999.9999
- **Minimum limit of size for stud** Q278: Minimum permissible diameter for the stud. Input range 0 to 99999.9999
- **Tolerance for center 1st axis** Q279: Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999
- **Tolerance for center 2nd axis** Q280: Permissible position deviation in the minor axis of the working plane. Input range 0 to 99999.9999



- **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
  0: No measuring log
  1: Create measuring log: By default, the TNC saves the **log file TCHPR422.TXT** in the directory in which your measuring program is stored.
  2: Interrupt program run and display the measuring log on the screen. Resume program run with NC Start.
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
  0 Do not interrupt program run, no error message
  1: Interrupt program run, output an error message
- **Tool for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring" on page 418). Input range: 0 to 32767.9, alternatively tool name with max. 16 characters
  0: Monitoring not active
  >0: Tool number in the tool table TOOL.T
- **No. of measuring points (4/3)** Q423: Specify whether the TNC should measure the stud with 4 or 3 probing points:
  4: Use 4 measuring points (standard setting)
  3: Use 3 measuring points
- **Type of traverse? Line=0/Arc=1** Q365: Definition of the path function with which the touch probe is to move between the measuring points if "traverse to clearance height" (Q301=1) is active.
  0: Move in a straight line between measuring points
  1: Move in a circular arc on the pitch circle diameter between measuring points

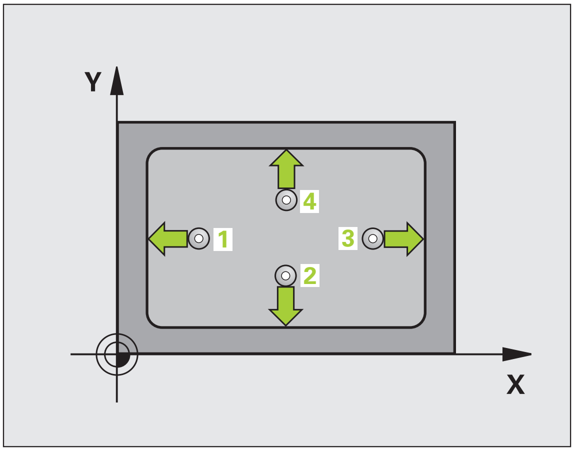
Example: NC blocks

```
5 TCH PROBE 422 MEAS. CIRCLE OUTSIDE
    Q273=+50  ;CENTER IN 1ST AXIS
    Q274=+50  ;CENTER IN 2ND AXIS
    Q262=75   ;NOMINAL DIAMETER
    Q325=+90  ;STARTING ANGLE
    Q247=+30  ;STEPPING ANGLE
    Q261=-5   ;MEASURING HEIGHT
    Q320=0    ;SET-UP CLEARANCE
    Q260=+10  ;CLEARANCE HEIGHT
    Q301=0    ;MOVE TO CLEARANCE
    Q277=35.15;MAX. LIMIT
    Q278=34.9 ;MIN. LIMIT
    Q279=0.05 ;TOLERANCE 1ST CENTER
    Q280=0.05 ;TOLERANCE 2ND CENTER
    Q281=1    ;MEASURING LOG
    Q309=0    ;PGM STOP IF ERROR
    Q330=0    ;TOOL
    Q423=4    ;NO. OF MEAS. POINTS
    Q365=1    ;TYPE OF TRAVERSE
```

16.7 MEAS. RECTAN. INSIDE (Cycle 423, DIN/ISO: G423)

Cycle run

Touch Probe Cycle 423 finds the center, length and width of a rectangular pocket. If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation values in system parameters.



1 Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe to the probe starting point **1** at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.
2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120).
3 Then the touch probe moves either paraxially at the measuring height or linearly at the clearance height to the next starting point **2** and probes the second touch point.
4 The TNC positions the touch probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
5 Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviations in the following Q parameters:

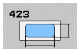
Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of length in the reference axis
Q155	Actual value of length in the minor axis
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q164	Deviation of side length in reference axis
Q165	Deviation of side length in minor axis

Please note while programming:

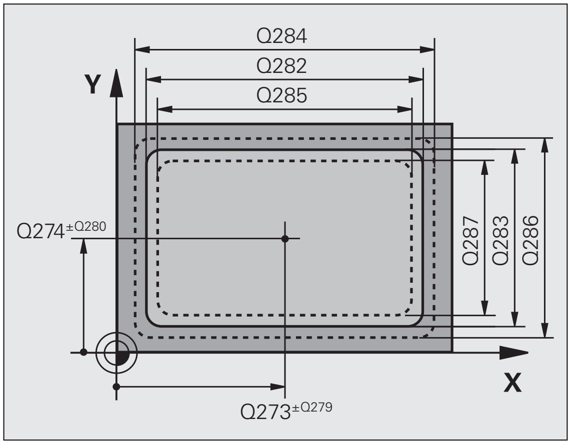
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

If the dimensions of the pocket and the safety clearance do not permit pre-positioning in the proximity of the touch points, the TNC always starts probing from the center of the pocket. In this case the touch probe does not return to the clearance height between the four measuring points.

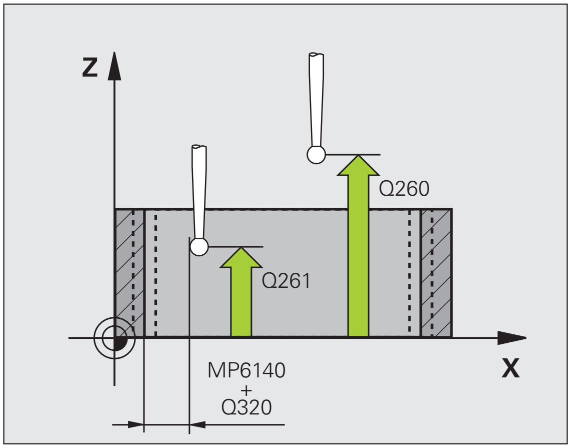
Cycle parameters



- **`Center in 1st axis`** Q273 (absolute): Center of the pocket in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **`Center in 2nd axis`** Q274 (absolute): Center of the pocket in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **`1 side length`** Q282: Pocket length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- **`2 side length`** Q283: Pocket length, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- **`Measuring height in the touch probe axis`** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999



- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **PREDEF**
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
  Alternative: **PREDEF**
- **Max. size limit 1st side length** Q284: Maximum permissible length of the pocket. Input range 0 to 99999.9999
- **Min. size limit 1st side length** Q285: Minimum permissible length of the pocket. Input range 0 to 99999.9999
- **Max. size limit 2nd side length** Q286: Maximum permissible width of the pocket. Input range 0 to 99999.9999
- **Min. size limit 2nd side length** Q287: Minimum permissible width of the pocket. Input range 0 to 99999.9999
- **Tolerance for center 1st axis** Q279: Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999
- **Tolerance for center 2nd axis** Q280: Permissible position deviation in the minor axis of the working plane. Input range 0 to 99999.9999



- **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
  0: No measuring log
  1: Create measuring log: By default, the TNC saves the **log file TCHPR423.TXT** in the directory in which your measuring program is stored.
  2: Interrupt program run and display the measuring log on the screen. Resume program run with NC Start.
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
  0 Do not interrupt program run, no error message
  1: Interrupt program run, output an error message
- **Tool for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring" on page 418). Input range: 0 to 32767.9, alternatively tool name with max. 16 characters
  0: Monitoring not active
  >0: Tool number in the tool table TOOL.T

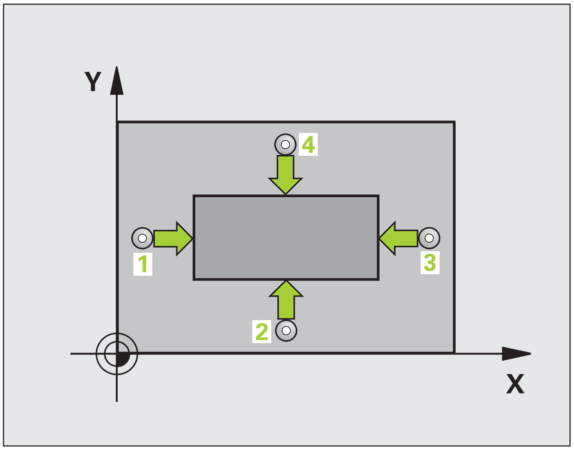
Example: NC blocks

```
5 TCH PROBE 423 MEAS. RECTAN. INSIDE
    Q273=+50  ;CENTER IN 1ST AXIS
    Q274=+50  ;CENTER IN 2ND AXIS
    Q282=80   ;1ST SIDE LENGTH
    Q283=60   ;2ND SIDE LENGTH
    Q261=-5   ;MEASURING HEIGHT
    Q320=0    ;SET-UP CLEARANCE
    Q260=+10  ;CLEARANCE HEIGHT
    Q301=1    ;MOVE TO CLEARANCE
    Q284=0    ;MAX. LIMIT 1ST SIDE
    Q285=0    ;MIN. LIMIT 1ST SIDE
    Q286=0    ;MAX. LIMIT 2ND SIDE
    Q287=0    ;MIN. LIMIT 2ND SIDE
    Q279=0    ;TOLERANCE 1ST CENTER
    Q280=0    ;TOLERANCE 2ND CENTER
    Q281=1    ;MEASURING LOG
    Q309=0    ;PGM STOP IF ERROR
    Q330=0    ;TOOL
```

16.8 MEASURE RECTANGLE OUTSIDE (Cycle 424, DIN/ISO: G424)

Cycle run

Touch Probe Cycle 424 finds the center, length and width of a rectangular stud. If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation values in system parameters.



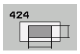
1. Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe to the probe starting point **1** at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.
2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120).
3. Then the touch probe moves either paraxially at the measuring height or linearly at the clearance height to the next starting point **2** and probes the second touch point.
4. The TNC positions the touch probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
5. Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviations in the following Q parameters:

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of length in the reference axis
Q155	Actual value of length in the minor axis
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q164	Deviation of side length in reference axis
Q165	Deviation of side length in minor axis

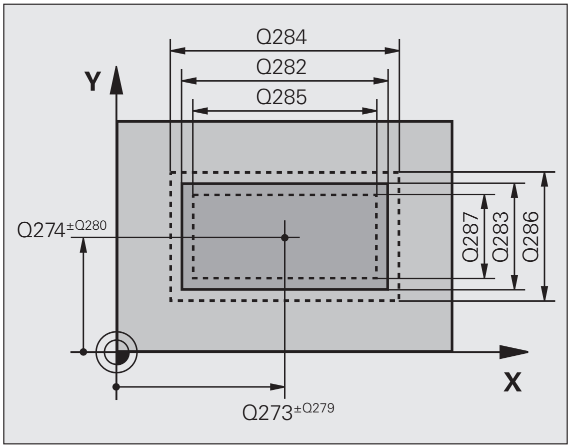
Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

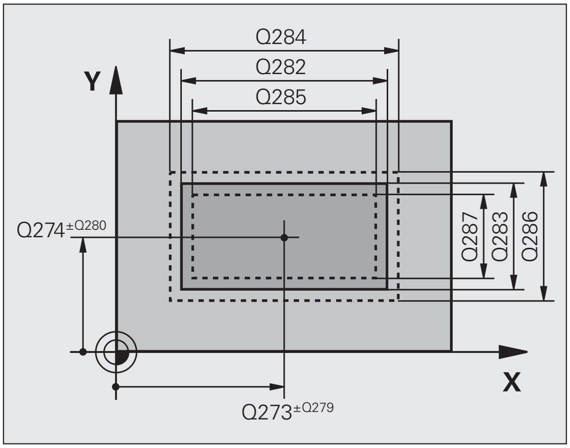
Cycle parameters

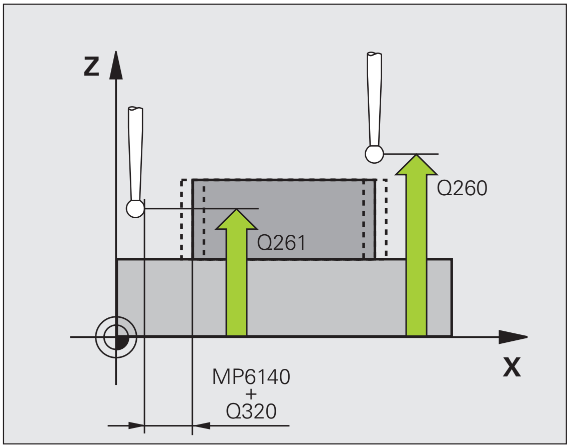


- **`Center in 1st axis`** Q273 (absolute): Center of the stud in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **`Center in 2nd axis`** Q274 (absolute): Center of the stud in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **`1 side length`** Q282: Stud length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- **`2 side length`** Q283: Stud length, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- **`Measuring height in the touch probe axis`** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999



- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **PREDEF**
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
  Alternative: **PREDEF**
- **Max. size limit 1st side length** Q284: Maximum permissible length of the stud. Input range 0 to 99999.9999
- **Min. size limit 1st side length** Q285: Minimum permissible length of the stud. Input range 0 to 99999.9999
- **Max. size limit 2nd side length** Q286: Maximum permissible width of the stud. Input range 0 to 99999.9999
- **Min. size limit 2nd side length** Q287: Minimum permissible width of the stud. Input range 0 to 99999.9999
- **Tolerance for center 1st axis** Q279: Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999
- **Tolerance for center 2nd axis** Q280: Permissible position deviation in the minor axis of the working plane. Input range 0 to 99999.9999





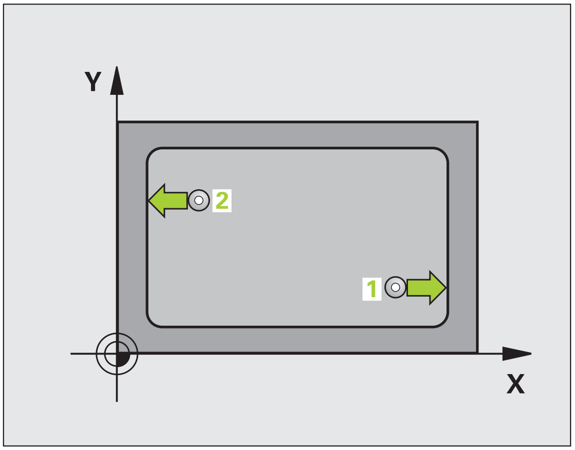
- **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
  0: No measuring log
  1: Create measuring log: By default, the TNC saves the **log file TCHPR424.TXT** in the directory in which your measuring program is stored.
  2: Interrupt program run and display the measuring log on the screen. Resume program run with NC Start.
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
  0 Do not interrupt program run, no error message
  1: Interrupt program run, output an error message
- **Tool for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring" on page 418). Input range: 0 to 32767.9, alternatively tool name with max. 16 characters:
  0: Monitoring not active
  >0: Tool number in the tool table TOOL.T

Example: NC blocks

```
5 TCH PROBE 424 MEAS. RECTAN. OUTS.
    Q273=+50  ;CENTER IN 1ST AXIS
    Q274=+50  ;CENTER IN 2ND AXIS
    Q282=75   ;1ST SIDE LENGTH
    Q283=35   ;2ND SIDE LENGTH
    Q261=-5   ;MEASURING HEIGHT
    Q320=0    ;SET-UP CLEARANCE
    Q260=+20  ;CLEARANCE HEIGHT
    Q301=0    ;MOVE TO CLEARANCE
    Q284=75.1 ;MAX. LIMIT 1ST SIDE
    Q285=74.9 ;MIN. LIMIT 1ST SIDE
    Q286=35   ;MAX. LIMIT 2ND SIDE
    Q287=34.95;MIN. LIMIT 2ND SIDE
    Q279=0.1  ;TOLERANCE 1ST CENTER
    Q280=0.1  ;TOLERANCE 2ND CENTER
    Q281=1    ;MEASURING LOG
    Q309=0    ;PGM STOP IF ERROR
    Q330=0    ;TOOL
```

16.9 MEASURE INSIDE WIDTH (Cycle 425, DIN/ISO: G425)

Cycle run



Touch Probe Cycle 425 measures the position and width of a slot (or pocket). If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in a system parameter.

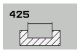
1 Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe to the probe starting point **1** at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.
2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120). The first probing is always in the positive direction of the programmed axis.
3 If you enter an offset for the second measurement, the TNC then moves the touch probe (if required, at clearance height) to the next starting point **2** and probes the second touch point. If the nominal length is large, the TNC moves the touch probe to the second touch point at rapid traverse. If you do not enter an offset, the TNC measures the width in the exact opposite direction.
4 Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviation in the following Q parameters:

Parameter number	Meaning
Q156	Actual value of measured length
Q157	Actual value of the centerline
Q166	Deviation of the measured length

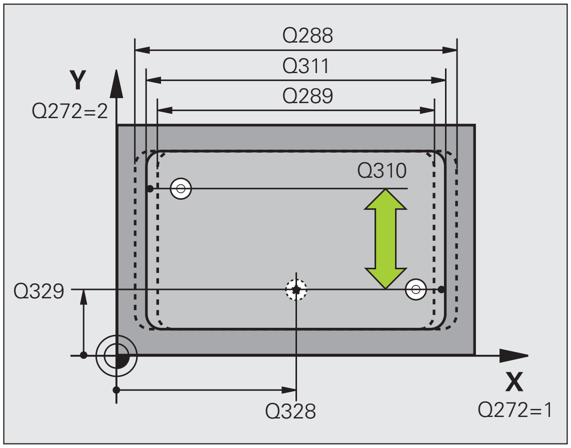
Please note while programming:

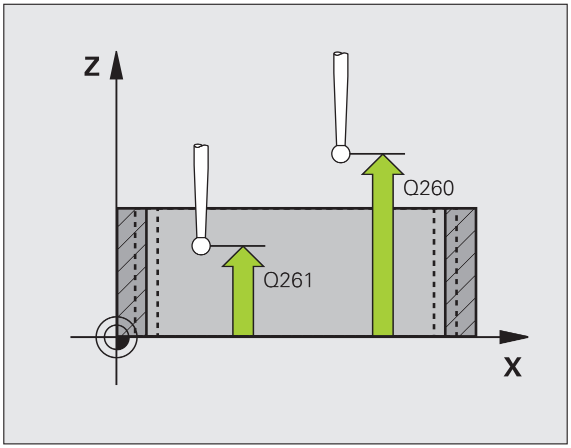
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **`Starting point in 1st axis`** Q328 (absolute): Starting point for probing in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **`Starting point in 2nd axis`** Q329 (absolute): Starting point for probing in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **`Offset for 2nd measurement`** Q310 (incremental): Distance by which the touch probe is displaced before the second measurement. If you enter 0, the TNC does not offset the touch probe. Input range -99999.9999 to 99999.9999
- **`Measuring axis`** Q272: Axis in the working plane in which the measurement is to be made:
  1:Reference axis = measuring axis
  2: Minor axis = measuring axis
- **`Measuring height in the touch probe axis`** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **`Clearance height`** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **`PREDEF`**
- **`Nominal length`** Q311: Nominal value of the length to be measured. Input range 0 to 99999.9999
- **`Maximum dimension`** Q288: Maximum permissible length. Input range 0 to 99999.9999
- **`Minimum dimension`** Q289: Minimum permissible length. Input range 0 to 99999.9999





- **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
  0: No measuring log
  1: Create measuring log: By default, the TNC saves the **log file TCHPR425.TXT** in the directory in which your measuring program is stored.
  2: Interrupt program run and display the measuring log on the screen. Resume program run with NC Start.
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
  0 Do not interrupt program run, no error message
  1: Interrupt program run, output an error message
- **Tool for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring" on page 418). Input range: 0 to 32767.9, alternatively tool name with max. 16 characters
  0: Monitoring not active
  >0: Tool number in the tool table TOOL.T
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
  Alternative: **PREDEF**

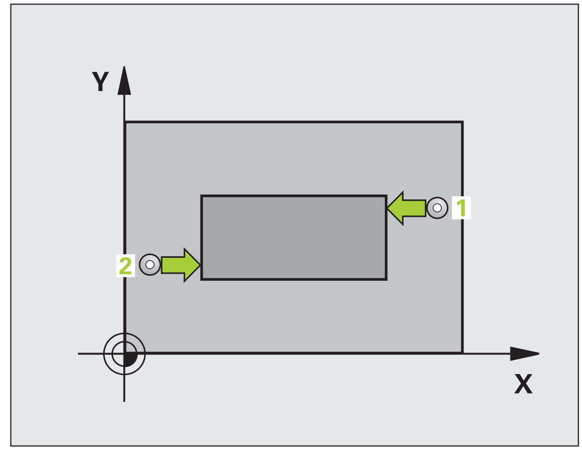
Example: NC blocks

```
5 TCH PROBE 425 MEASURE INSIDE WIDTH
    Q328=+75   ;STARTNG PNT 1ST AXIS
    Q329=-12.5;STARTNG PNT 2ND AXIS
    Q310=+0    ;OFFS. 2ND MEASUREMENT
    Q272=1     ;MEASURING AXIS
    Q261=-5    ;MEASURING HEIGHT
    Q260=+10   ;CLEARANCE HEIGHT
    Q311=25    ;NOMINAL LENGTH
    Q288=25.05;MAX. LIMIT
    Q289=25    ;MIN. LIMIT
    Q281=1     ;MEASURING LOG
    Q309=0     ;PGM STOP IF ERROR
    Q330=0     ;TOOL
    Q320=0     ;SET-UP CLEARANCE
    Q301=0     ;MOVE TO CLEARANCE
```

16.10 MEASURE RIDGE WIDTH (Cycle 426, DIN/ISO: G426)

Cycle run

Touch Probe Cycle 426 measures the position and width of a ridge. If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.



1. Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe to the probe starting point **1** at rapid traverse (value from MP6150). The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.
2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120). The first probing is always in the negative direction of the programmed axis.
3. Then the touch probe moves at clearance height to the next starting position and probes the second touch point.
4. Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviation in the following Q parameters:

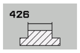
Parameter number	Meaning
Q156	Actual value of measured length
Q157	Actual value of the centerline
Q166	Deviation of the measured length

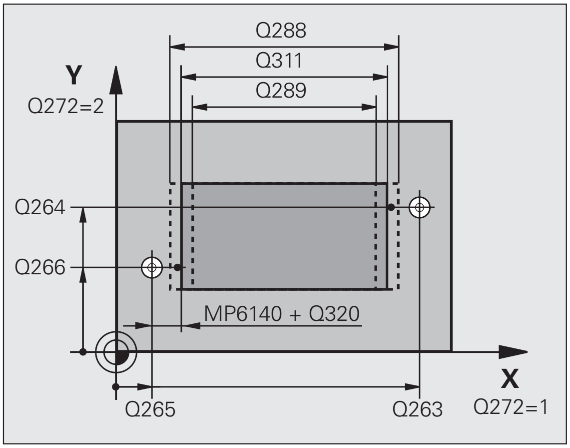
Please note while programming:

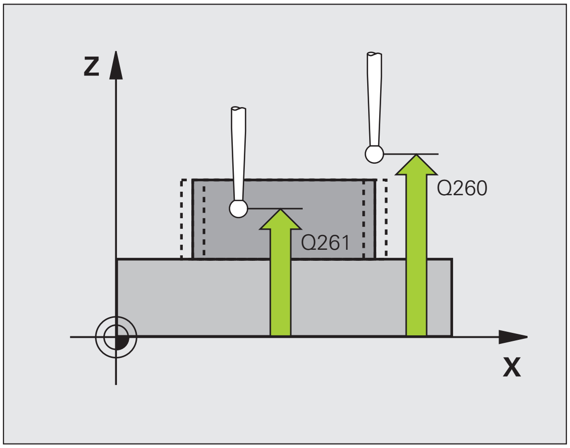
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Ensure that the first measurement is always carried out in the negative direction of the selected measuring axis. Define **Q263** and **Q264** correspondingly.

Cycle parameters







- **1st meas. point 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **1st meas. point 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 1st axis** Q265 (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **2nd meas. point 2nd axis** Q266 (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Measuring axis** Q272: Axis in the working plane in which the measurement is to be made:
  1:Reference axis = measuring axis
  2: Minor axis = measuring axis
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **PREDEF**
- **Nominal length** Q311: Nominal value of the length to be measured. Input range 0 to 99999.9999
- **Maximum dimension** Q288: Maximum permissible length. Input range 0 to 99999.9999
- **Minimum dimension** Q289: Minimum permissible length. Input range 0 to 99999.9999

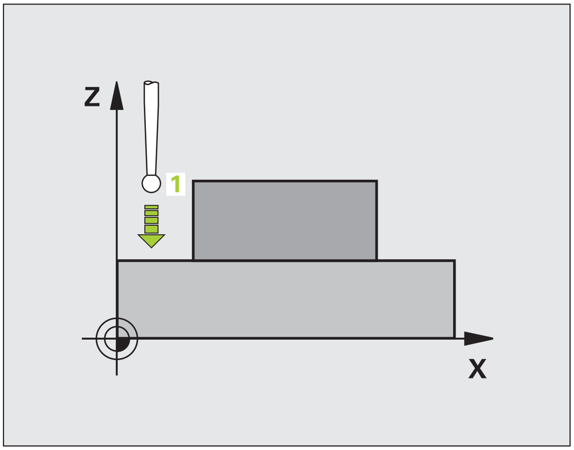
- **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
  0: No measuring log
  1: Create measuring log: By default, the TNC saves the **log file TCHPR426.TXT** in the directory in which your measuring program is stored.
  2: Interrupt program run and display the measuring log on the screen. Resume program run with NC Start.
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
  0 Do not interrupt program run, no error message
  1: Interrupt program run, output an error message
- **Tool for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring" on page 418). Input range: 0 to 32767.9, alternatively tool name with max. 16 characters
  0: Monitoring not active
  >0: Tool number in the tool table TOOL.T

Example: NC blocks

```
5 TCH PROBE 426 MEASURE RIDGE WIDTH
    Q263=+50  ;1ST POINT 1ST AXIS
    Q264=+25  ;1ST POINT 2ND AXIS
    Q265=+50  ;2ND POINT 1ST AXIS
    Q266=+85  ;2ND POINT 2ND AXIS
    Q272=2    ;MEASURING AXIS
    Q261=-5   ;MEASURING HEIGHT
    Q320=0    ;SET-UP CLEARANCE
    Q260=+20  ;CLEARANCE HEIGHT
    Q311=45   ;NOMINAL LENGTH
    Q288=45   ;MAX. LIMIT
    Q289=44.95;MIN. LIMIT
    Q281=1    ;MEASURING LOG
    Q309=0    ;PGM STOP IF ERROR
    Q330=0    ;TOOL
```

16.11 MEASURE COORDINATE (Cycle 427, DIN/ISO: G427)

Cycle run



Touch Probe Cycle 427 finds a coordinate in a selectable axis and saves the value in a system parameter. If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.

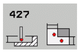
1 Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe to the probe starting point **1** at rapid traverse (value from MP6150). The TNC offsets the touch probe by the safety clearance in the direction opposite to the defined traverse direction.
2 Then the TNC positions the touch probe to the entered touch point **1** in the working plane and measures the actual value in the selected axis.
3 Finally the TNC returns the touch probe to the clearance height and saves the measured coordinate in the following Q parameter:

Parameter number	Meaning
Q160	Measured coordinate

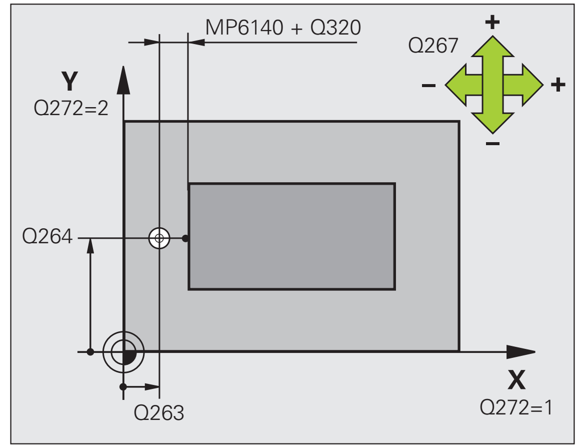
Please note while programming:

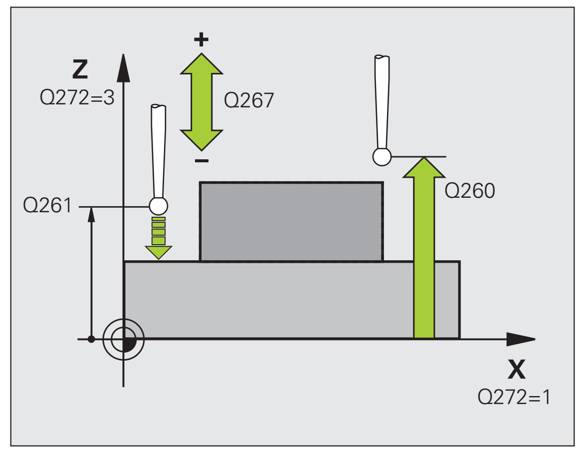
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **`1st meas. point 1st axis`** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **`1st meas. point 2nd axis`** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **`Measuring height in the touch probe axis`** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **`Set-up clearance`** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **`PREDEF`**
- **`Measuring axis (1 to 3: 1=reference axis)`** Q272: Axis in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
  3: Touch probe axis = measuring axis
- **`Traverse direction 1`** Q267: Direction in which the touch probe is to approach the workpiece:
  –1: Negative traverse direction
  +1: Positive traverse direction
- **`Clearance height`** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **`PREDEF`**





- **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
  0: No measuring log
  1: Create measuring log: By default, the TNC saves the **log file TCHPR427.TXT** in the directory in which your measuring program is stored.
  2: Interrupt program run and display the measuring log on the screen. Resume program run with NC Start.
- **Maximum limit of size** Q288: Maximum permissible measured value. Input range -99999.9999 to 99999.9999
- **Minimum limit of size** Q289: Minimum permissible measured value. Input range -99999.9999 to 99999.9999
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
  0 Do not interrupt program run, no error message
  1: Interrupt program run, output an error message
- **Tool for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring" on page 418). Input range: 0 to 32767.9, alternatively tool name with max. 16 characters:
  0: Monitoring not active
  >0: Tool number in the tool table TOOL.T

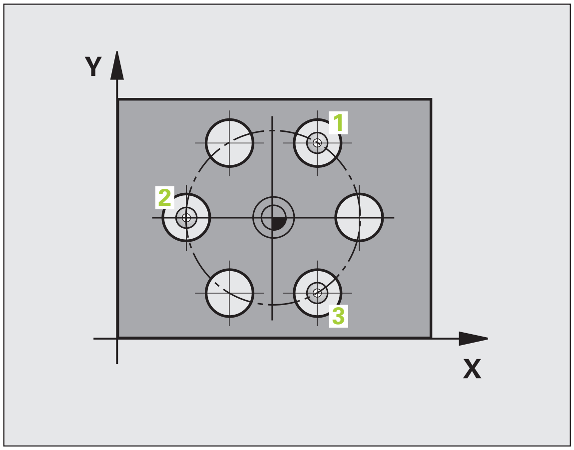
Example: NC blocks

```
5 TCH PROBE 427 MEASURE COORDINATE
    Q263=+35  ;1ST POINT 1ST AXIS
    Q264=+45  ;1ST POINT 2ND AXIS
    Q261=+5   ;MEASURING HEIGHT
    Q320=0    ;SET-UP CLEARANCE
    Q272=3    ;MEASURING AXIS
    Q267=-1   ;TRAVERSE DIRECTION
    Q260=+20  ;CLEARANCE HEIGHT
    Q281=1    ;MEASURING LOG
    Q288=5.1  ;MAX. LIMIT
    Q289=4.95 ;MIN. LIMIT
    Q309=0    ;PGM STOP IF ERROR
    Q330=0    ;TOOL
```

16.12 MEASURE BOLT HOLE CIRCLE (Cycle 430, DIN/ISO: G430)

Cycle run

Touch Probe Cycle 430 finds the center and diameter of a bolt hole circle by probing three holes. If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.



1 Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe at rapid traverse (value from MP6150) to the point entered as center of the first hole **1**
2 Then the touch probe moves to the entered measuring height and probes four points to find the first hole center.
3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
4 The TNC moves the touch probe to the entered measuring height and probes four points to find the second hole center.
5 The touch probe returns to the clearance height and then to the position entered as center of the third hole **3**.
6 The TNC moves the touch probe to the entered measuring height and probes four points to find the third hole center.
7 Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviations in the following Q parameters:

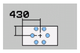
Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of bolt hole circle diameter
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q163	Deviation of bolt hole circle diameter

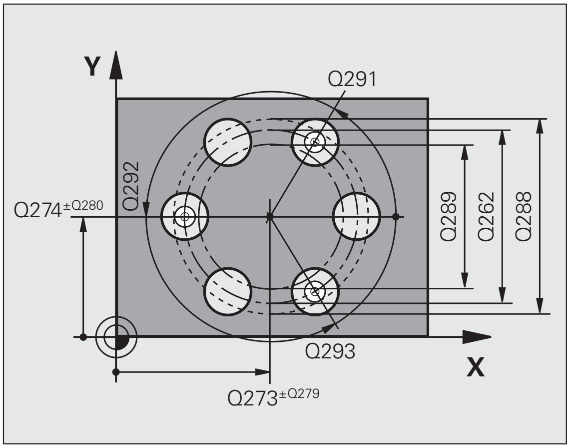
Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle 430 only monitors for tool breakage; there is no automatic tool compensation.

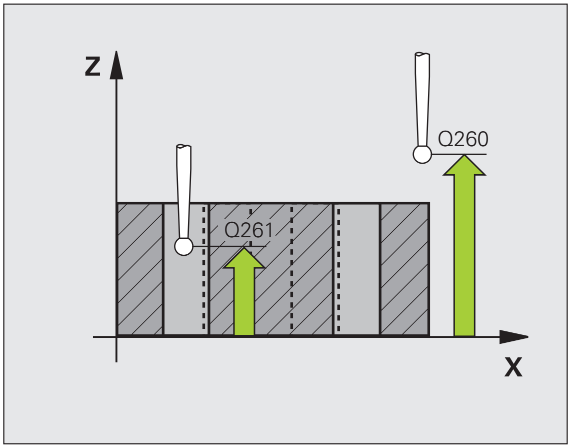
Cycle parameters





- **Center in 1st axis** Q273 (absolute): Bolt hole circle center (nominal value) in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Center in 2nd axis** Q274 (absolute): Bolt hole circle center (nominal value) in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Nominal diameter** Q262: Enter the bolt hole circle diameter. Input range 0 to 99999.9999
- **Angle of 1st hole** Q291 (absolute): Polar coordinate angle of the first hole center in the working plane. Input range -360.0000 to 360.0000
- **Angle of 2nd hole** Q292 (absolute): Polar coordinate angle of the second hole center in the working plane. Input range -360.0000 to 360.0000
- **Angle of 3rd hole** Q293 (absolute): Polar coordinate angle of the third hole center in the working plane. Input range -360.0000 to 360.0000

- **`Measuring height in the touch probe axis`** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range -99999.9999 to 99999.9999
- **`Clearance height`** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **`PREDEF`**
- **`Maximum limit of size`** Q288: Maximum permissible diameter of bolt hole circle. Input range 0 to 99999.9999
- **`Minimum limit of size`** Q289: Minimum permissible diameter of bolt hole circle. Input range 0 to 99999.9999
- **`Tolerance for center 1st axis`** Q279: Permissible position deviation in the reference axis of the working plane. Input range 0 to 99999.9999
- **`Tolerance for center 2nd axis`** Q280: Permissible position deviation in the minor axis of the working plane. Input range 0 to 99999.9999



- **`Measuring log`** Q281: Definition of whether the TNC is to create a measuring log:
  0: No measuring log
  1: Create measuring log: By default, the TNC saves the **log file TCHPR430.TXT** in the directory in which your measuring program is stored.
  2: Interrupt program run and display the measuring log on the screen. Resume program run with NC Start.
- **`PGM stop if tolerance error`** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt program run and output an error message:
  0 Do not interrupt program run, no error message
  1: Interrupt program run, output an error message
- **`Tool number for monitoring`** Q330: Definition of whether the TNC is to monitor for tool breakage (see "Tool monitoring" on page 418). Input range 0 to 32767.9; alternatively tool name with max. 16 characters
  0: Monitoring not active
  >0: Tool number in the tool table TOOL.T

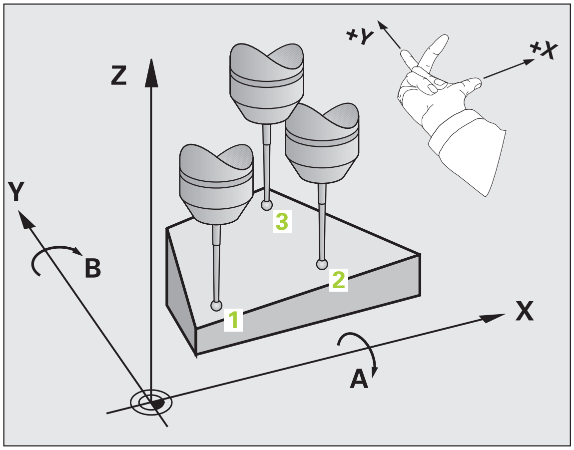
Example: NC blocks

```
5 TCH PROBE 430 MEAS. BOLT HOLE CIRC
    Q273=+50  ;CENTER IN 1ST AXIS
    Q274=+50  ;CENTER IN 2ND AXIS
    Q262=80   ;NOMINAL DIAMETER
    Q291=+0   ;ANGLE OF 1ST HOLE
    Q292=+90  ;ANGLE OF 2ND HOLE
    Q293=+180 ;ANGLE OF 3RD HOLE
    Q261=-5   ;MEASURING HEIGHT
    Q260=+10  ;CLEARANCE HEIGHT
    Q288=80.1 ;MAX. LIMIT
    Q289=79.9 ;MIN. LIMIT
    Q279=0.15 ;TOLERANCE 1ST CENTER
    Q280=0.15 ;TOLERANCE 2ND CENTER
    Q281=1    ;MEASURING LOG
    Q309=0    ;PGM STOP IF ERROR
    Q330=0    ;TOOL
```

16.13 MEASURE PLANE (Cycle 431, DIN/ISO: G431)

Cycle run

Touch Probe Cycle 431 finds the angles of a plane by measuring three points. It saves the measured values in system parameters.



1. Following the positioning logic (see "Executing touch probe cycles" on page 336), the TNC positions the touch probe at rapid traverse (value from MP6150) to the programmed starting point **1** and measures the first touch point of the plane. The TNC offsets the touch probe by the safety clearance in the direction opposite to the direction of probing.
2. The touch probe returns to the clearance height and then moves in the working plane to starting point **2** and measures the actual value of the second touch point of the plane.
3. The touch probe returns to the clearance height and then moves in the working plane to starting point **3** and measures the actual value of the third touch point of the plane.
4. Finally the TNC returns the touch probe to the clearance height and saves the measured angle values in the following Q parameters:

Parameter number	Meaning
Q158	Projection angle of the A axis
Q159	Projection angle of the B axis
Q170	Spatial angle A
Q171	Spatial angle B
Q172	Spatial angle C
Q173 to Q175	Measured values in the touch probe axis (first to third measurement)

Please note while programming:

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

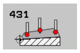
For the TNC to be able to calculate the angular values, the three measuring points must not be positioned on one straight line.

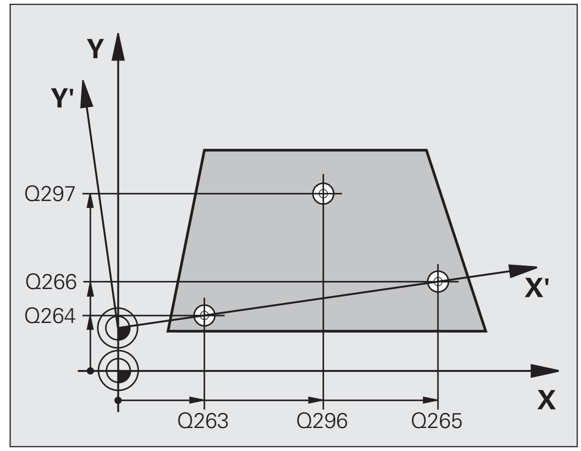
The spatial angles that are needed for tilting the working plane are saved in parameters Q170 – Q172. With the first two measuring points you also specify the direction of the reference axis when tilting the working plane.

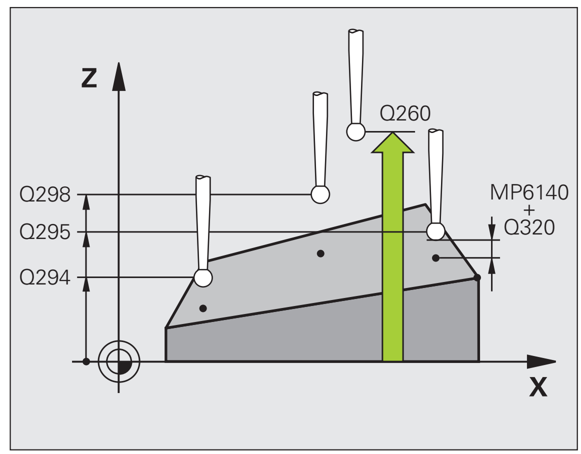
The third measuring point determines the direction of the tool axis. Define the third measuring point in the direction of the positive Y axis to ensure that the position of the tool axis in a clockwise coordinate system is correct.

If you run the cycle while a tilted working plane is active, the spatial angle is measured with respect to the tilted coordinate system. In this case, use the measured spatial angles with **PLANE RELATIV.**

Cycle parameters







- **`1st meas. point 1st axis`** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **`1st meas. point 2nd axis`** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **`1st meas. point 3rd axis`** Q294 (absolute): Coordinate of the first touch point in the touch probe axis. Input range -99999.9999 to 99999.9999
- **`2nd meas. point 1st axis`** Q265 (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **`2nd meas. point 2nd axis`** Q266 (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **`2nd meas. point 3rd axis`** Q295 (absolute): Coordinate of the second touch point in the touch probe axis. Input range -99999.9999 to 99999.9999
- **`3rd meas. point 1st axis`** Q296 (absolute): Coordinate of the third touch point in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **`3rd meas. point 2nd axis`** Q297 (absolute): Coordinate of the third touch point in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **`3rd meas. point 3rd axis`** Q298 (absolute): Coordinate of the third touch point in the touch probe axis. Input range -99999.9999 to 99999.9999

- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur. Input range -99999.9999 to 99999.9999; alternatively **PREDEF**
- **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
  0: No measuring log
  1: Create measuring log: By default, the TNC saves the **log file TCHPR431.TXT** in the directory in which your measuring program is stored.
  2: Interrupt program run and display the measuring log on the screen. Resume program run with NC Start.

Example: NC blocks

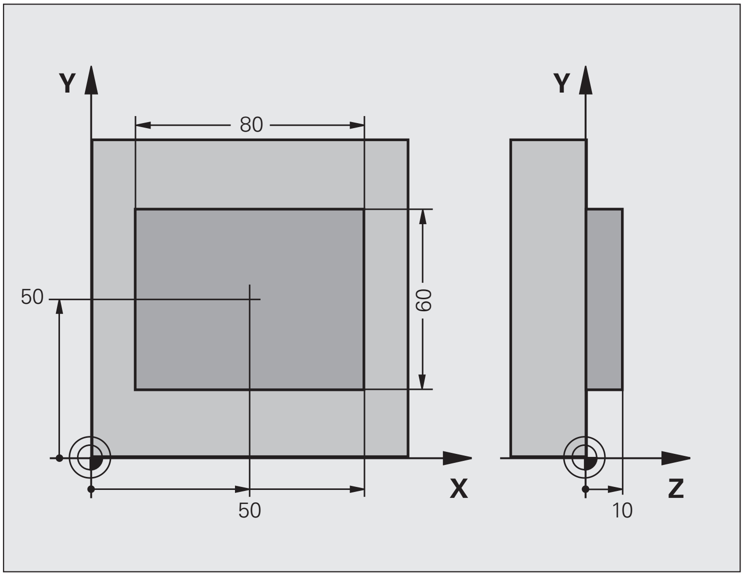
```
5 TCH PROBE 431 MEASURE PLANE
    Q263=+20  ;1ST POINT 1ST AXIS
    Q264=+20  ;1ST POINT 2ND AXIS
    Q294=+10  ;1ST POINT 3RD AXIS
    Q265=+90  ;2ND POINT 1ST AXIS
    Q266=+25  ;2ND POINT 2ND AXIS
    Q295=+15  ;2ND POINT 3RD AXIS
    Q296=+50  ;3RD POINT 1ST AXIS
    Q297=+80  ;3RD POINT 2ND AXIS
    Q298=+20  ;3RD POINT 3RD AXIS
    Q320=0    ;SET-UP CLEARANCE
    Q260=+5   ;CLEARANCE HEIGHT
    Q281=1    ;MEASURING LOG
```

16.14 Programming examples

Example: Measuring and reworking a rectangular stud

Program sequence:

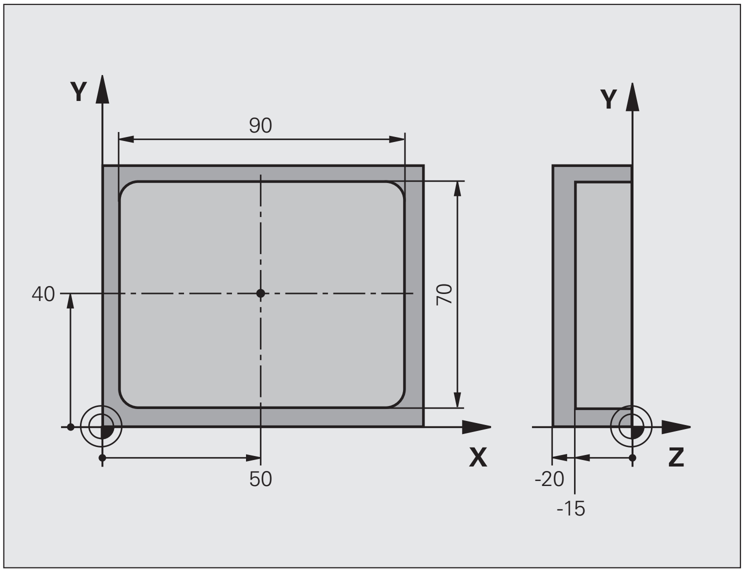
- Roughing with 0.5 mm finishing allowance
- Measuring
- Rectangular stud finishing in accordance with the measured values



`0 BEGIN PGM BEAMS MM`	
`1 TOOL CALL 69 Z`	Tool call for roughing
`2 L Z+100 R0 FMAX`	Retract the tool
`3 FN 0: Q1 = +81`	Pocket length in X (roughing dimension)
`4 FN 0: Q2 = +61`	Pocket length in Y (roughing dimension)
`5 CALL LBL 1`	Call subprogram for machining
`6 L Z+100 R0 FMAX`	Retract the tool, change the tool
`7 TOOL CALL 99 Z`	Call the touch probe
`8 TCH PROBE 424 MEAS. RECTAN. OUTS.`	Measure the rough-milled rectangle
`Q273=+50 ;CENTER IN 1ST AXIS`	
`Q274=+50 ;CENTER IN 2ND AXIS`	
`Q282=80 ;1ST SIDE LENGTH`	Nominal length in X (final dimension)
`Q283=60 ;2ND SIDE LENGTH`	Nominal length in Y (final dimension)
`Q261=-5 ;MEASURING HEIGHT`	
`Q320=0 ;SET-UP CLEARANCE`	
`Q260=+30 ;CLEARANCE HEIGHT`	
`Q301=0 ;MOVE TO CLEARANCE`	
`Q284=0 ;MAX. LIMIT 1ST SIDE`	Input values for tolerance checking not required

Q285=0 ;MIN. LIMIT 1ST SIDE	
Q286=0 ;MAX. LIMIT 2ND SIDE	
Q287=0 ;MIN. LIMIT 2ND SIDE	
Q279=0 ;TOLERANCE 1ST CENTER	
Q280=0 ;TOLERANCE 2ND CENTER	
Q281=0 ;MEASURING LOG	No measuring log transmission
Q309=0 ;PGM STOP IF ERROR	Do not output an error message
Q330=0 ;TOOL NUMBER	No tool monitoring
9 FN 2: Q1 = +Q1 - +Q164	Calculate length in X including the measured deviation
10 FN 2: Q2 = +Q2 - +Q165	Calculate length in Y including the measured deviation
11 L Z+100 R0 FMAX	Retract the touch probe, change the tool
12 TOOL CALL 1 Z S5000	Tool call for finishing
13 CALL LBL 1	Call subprogram for machining
14 L Z+100 R0 FMAX M2	Retract the tool, end program
15 LBL 1	Subprogram with fixed cycle for rectangular stud
16 CYCL DEF 213 STUD FINISHING	
Q200=20 ;SET-UP CLEARANCE	
Q201=-10 ;DEPTH	
Q206=150 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q207=500 ;FEED RATE FOR MILLING	
Q203=+10 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q216=+50 ;CENTER IN 1ST AXIS	
Q217=+50 ;CENTER IN 2ND AXIS	
Q218=Q1 ;FIRST SIDE LENGTH	Length in X variable for roughing and finishing
Q219=Q2 ;SECOND SIDE LENGTH	Length in Y variable for roughing and finishing
Q220=0 ;CORNER RADIUS	
Q221=0 ;ALLOWANCE IN 1ST AXS	
17 CYCL CALL M3	Cycle call
18 LBL 0	End of subprogram
19 END PGM BEAMS MM	

Example: Measuring a rectangular pocket and recording the results



0 BEGIN PGM BSMEAS MM	
1 TOOL CALL 1 Z	Tool call for touch probe
2 L Z+100 R0 FMAX	Retract the touch probe
3 TCH PROBE 423 MEAS. RECTAN. INSIDE	
Q273=+50 ;CENTER IN 1ST AXIS	
Q274=+40 ;CENTER IN 2ND AXIS	
Q282=90 ;1ST SIDE LENGTH	Nominal length in X
Q283=70 ;2ND SIDE LENGTH	Nominal length in Y
Q261=-5 ;MEASURING HEIGHT	
Q320=0 ;SET-UP CLEARANCE	
Q260=+20 ;CLEARANCE HEIGHT	
Q301=0 ;MOVE TO CLEARANCE	

`Q284=90.15;MAX. LIMIT 1ST SIDE`	Maximum limit in X
`Q285=89.95;MIN. LIMIT 1ST SIDE`	Minimum limit in X
`Q286=70.1 ;MAX. LIMIT 2ND SIDE`	Maximum limit in Y
`Q287=69.9 ;MIN. LIMIT 2ND SIDE`	Minimum limit in Y
`Q279=0.15 ;TOLERANCE 1ST CENTER`	Permissible position deviation in X
`Q280=0.1  ;TOLERANCE 2ND CENTER`	Permissible position deviation in Y
`Q281=1    ;MEASURING LOG`	Save measuring log to a file
`Q309=0    ;PGM STOP IF ERROR`	Do not display an error message in case of a tolerance violation
`Q330=0    ;TOOL NUMBER`	No tool monitoring
`4 L Z+100 R0 FMAX M2`	Retract the tool, end program
`5 END PGM BSMEAS  MM`	

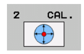
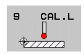
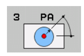
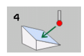
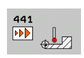
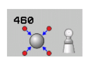
17

Touch Probe Cycles: Special Functions

17.1 Fundamentals

Overview

The TNC provides seven cycles for the following special purposes:

Cycle	Soft key	Page
2 CALIBRATE TS Radius calibration of the touch trigger probe		Page 465
9 CALIBRATE TS LENGTH Length calibration of the touch trigger probe		Page 466
3 MEASURING Cycle for defining OEM cycles		Page 467
4 MEASURING IN 3-D Measuring cycle for 3-D probing for defining OEM cycles		Page 469
440 MEASURE AXIS SHIFT		Page 471
441 FAST PROBING		Page 474
460 CALIBRATE TS Radius and length calibration on a calibration sphere		Page 476

17.2 CALIBRATE TS (Cycle 2)

Cycle run

Touch Probe Cycle 2 automatically calibrates a touch trigger probe using a ring gauge or a precision stud as calibration standard.

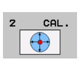
1. The touch probe moves at rapid traverse (value from MP6150) to the clearance height (but only if the current position is below the clearance height).
2. Then the TNC positions the touch probe in the working plane to the center of the ring gauge (calibration from inside) or in its proximity (calibration from outside).
3. The touch probe then moves to the measuring depth (result of MP618x.2 and MP6185.x) and probes the ring gauge successively in X+, Y+, X- and Y-.
4. Finally, the TNC moves the touch probe to the clearance height and writes the effective radius of the ball tip to the calibration data.

Please note while programming:

Before you begin calibrating, you must define in Machine Parameters 6180.0 to 6180.2 the center of the calibrating workpiece in the working space of the machine (REF coordinates).

If you are working with several traverse ranges you can save a separate set of coordinates for the center of each calibrating workpiece (MP6181.1 to 6181.2 and MP6182.1 to 6182.2).

Cycle parameters



- **`Clearance height`** (absolute): Coordinate in the touch probe axis at which the touch probe cannot collide with the calibration workpiece or any fixtures. Input range -99999.9999 to 99999.9999
- **`Radius of ring gauge`**: Radius of the calibrating workpiece. Input range 0 to 99999.9999
- **`Inside calib. =0/outs. calib.=1`**: Definition of whether the TNC is to calibrate from inside or outside:
  0: Calibrate from inside
  1: Calibrate from outside

Example: NC blocks

```
5 TCH PROBE 2.0 CALIBRATE TS
```

```
6 TCH PROBE
2.1 HEIGHT: +50 R +25.003 DIRECTION: 0
```

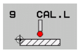
17.3 CALIBRATE TS LENGTH (Cycle 9)

Cycle run

Touch Probe Cycle 9 automatically calibrates the length of a touch trigger probe at a point that you determine.

1. Pre-position the touch probe so that the touch probe axis coordinate defined in the cycle can be accessed without collision.
2. The TNC moves the touch probe in the direction of the negative tool axis until a trigger signal is released.
3. Finally, the TNC moves the touch probe back to the starting point of the probing process and writes the effective touch probe length to the calibration data.

Cycle parameters



- **`Coordinate of datum`** (absolute): Exact coordinate of the point that is to be probed. Input range -99999.9999 to 99999.9999
- **`Reference system? (0=ACT/1=REF)`**: Specify the coordinate system on which the entered datum is to be based:
  0: Entered datum is based on the active workpiece coordinate system (ACT system)
  1: Entered datum is based on the active machine coordinate system (REF system)

Example: NC blocks

```
5 L X-235 Y+356 R0 FMAX
6 TCH PROBE 9.0 CALIBRATE TS LENGTH
7 TCH PROBE 9.1 DATUM +50 REFERENCE
SYSTEM 0
```

17.4 MEASURING (Cycle 3)

Cycle run

Touch Probe Cycle 3 measures any position on the workpiece in a selectable direction. Unlike other measuring cycles, Cycle 3 enables you to enter the measuring range **SET UP** and feed rate **F** directly. Also, the touch probe retracts by the definable value **MB** after determining the measured value.

1 The touch probe moves from the current position at the entered feed rate in the defined probing direction. The probing direction must be defined in the cycle as a polar angle.

2 After the TNC has saved the position, the touch probe stops. The TNC saves the X, Y, Z coordinates of the probe-tip center in three successive Q parameters. The TNC does not conduct any length or radius compensations. You define the number of the first result parameter in the cycle.

3 Finally, the TNC moves the touch probe back by that value against the probing direction that you defined in the parameter **MB**.

Please note while programming:

The exact behavior of Touch Probe Cycle 3 is defined by your machine tool builder or a software manufacturer who uses it within specific touch probe cycles.

MP6130 (maximum traverse to touch point) and MP6120 (probing feed rate), which are effective in other measuring cycles, do not apply in touch probe cycle 3.

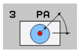
Remember that the TNC always writes to four successive Q-parameters.

If the TNC was not able to determine a valid touch point, the program is run without error message. In this case the TNC assigns the value –1 to the 4th result parameter so that you can deal with the error yourself.

The TNC retracts the touch probe by no more than the retraction distance **MB** and does not pass the starting point of the measurement. This rules out any collision during retraction.

With function **FN17: SYSWRITE ID 990 NR 6** you can set whether the cycle runs through the probe input X12 or X13.

Cycle parameters



- **Parameter number for result**: Enter the number of the Q parameter to which you want the TNC to assign the first measured coordinate (X). The values Y and Z are in the immediately following Q parameters. Input range: 0 to 1999
- **Probing axis**: Enter the axis in whose direction the probe is to move and confirm with the ENT key. Input range: X, Y or Z
- **Probing angle**: Angle, measured from the defined **probing axis**, in which the touch probe is to move. Confirm with ENT. Input range –180.0000 to 180.0000
- **Maximum measuring range**: Enter the maximum distance from the starting point by which the touch probe is to move. Confirm with ENT. Input range -99999.9999 to 99999.9999
- **Feed rate for measurement**: Enter the measuring feed rate in mm/min. Input range 0 to 3000.000
- **Maximum retraction distance**: Traverse path in the direction opposite the probing direction, after the stylus was deflected. The TNC returns the touch probe to a point no farther than the starting point, so that there can be no collision. Input range 0 to 99999.9999
- **Reference system? (0=ACT/1=REF)**: Specify whether the probing direction and the result of measurement are to be referenced to the actual coordinate system (**ACT**, can be shifted or rotated), or to the machine coordinate system (**REF**):
  0: Probe in the current system and save measurement result in the **ACT** system
  1: Probe in the machine-based REF system and save measurement result in the **REF** system
- **Error mode (0=OFF/1=ON)**: Specify whether the TNC is to issue an error message if the stylus is deflected at cycle start. If mode **1** is selected, the TNC saves the value **2.0** in the 4th result parameter and continues the cycle:
  0: Issue error message
  1: Do not issue error message

Example: NC blocks

```
4 TCH PROBE 3.0 MEASURING
5 TCH PROBE 3.1 Q1
6 TCH PROBE 3.2 X ANGLE: +15
7 TCH PROBE 3.3 DIST +10 F100 MB1
 REFERENCE SYSTEM:0
8 TCH PROBE 3.4 ERRORMODE1
```

17.5 MEASURING IN 3-D (Cycle 4, FCL 3 function)

Cycle run

Cycle 4 is an auxiliary cycle that you can only use in conjunction with external software! The TNC does not provide any cycle with which you can calibrate the touch probe.

Touch probe cycle 4 measures any position on the workpiece in the probing direction defined by a vector. Unlike other measuring cycles, Cycle 4 enables you to enter the measuring distance and feed rate directly. Also, the touch probe retracts by a definable value after determining the measured value.

1. The touch probe moves from the current position at the entered feed rate in the defined probing direction. Define the probing direction in the cycle by using a vector (delta values in X, Y and Z).
2. After the TNC has saved the position, the touch probe stops. The TNC saves the X, Y, Z coordinates of the probe-tip center (without calculation of the calibration data) in three successive Q parameters. You define the number of the first parameter in the cycle.
3. Finally, the TNC moves the touch probe back by that value against the probing direction that you defined in the parameter **MB.**

Please note while programming:

The TNC retracts the touch probe by no more than the retraction distance **MB** and does not pass the starting point of the measurement. This rules out any collision during retraction.

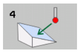
Ensure during pre-positioning that the TNC moves the probe-tip center without compensation to the defined position!

Remember that the TNC always writes to 4 successive Q parameters. If the TNC could not determine a valid touch point, the 4th result parameter will have the value –1.

The TNC saves the measured values without calculating the calibration data of the touch probe.

With function **FN17: SYSWRITE ID 990 NR 6** you can set whether the cycle runs through the probe input X12 or X13.

Cycle parameters



- **`Parameter number for result`**: Enter the number of the Q parameter to which you want the TNC to assign the first coordinate (X). Input range 0 to 1999
- **`Relative measuring path in X`**: X component of the direction vector defining the direction in which the touch probe is to move. Input range -99999.9999 to 99999.9999
- **`Relative measuring path in Y`**: Y component of the direction vector defining the direction in which the touch probe is to move. Input range -99999.9999 to 99999.9999
- **`Relative measuring path in Z`**: Z component of the direction vector defining the direction in which the touch probe is to move. Input range -99999.9999 to 99999.9999
- **`Maximum measuring path`**: Enter the maximum distance from the starting point by which the touch probe may move along the direction vector. Input range -99999.9999 to 99999.9999
- **`Feed rate for measurement`**: Enter the measuring feed rate in mm/min. Input range 0 to 3000.000
- **`Maximum retraction distance`**: Traverse path in the direction opposite the probing direction, after the stylus was deflected. Input range 0 to 99999.9999
- **`Reference system? (0=ACT/1=REF)`**: Specify whether the result of measurement is to be saved in the actual coordinate system (**ACT**, can be shifted or rotated), or with respect to the machine coordinate system (**REF**).
  0: Save the measurement result in the **ACT** system
  1: Save the measurement result in the **REF** system

Example: NC blocks

```
5 TCH PROBE 4.0 MEASURING IN 3-D
6 TCH PROBE 4.1 Q1
7 TCH PROBE 4.2 IX-0.5 IY-1 IZ-1
8 TCH PROBE
4.3 DIST +45 F100 MB50 REFERENCE SYSTEM:0
```

17.6 MEASURE AXIS SHIFT (Touch Probe Cycle 440, DIN/ISO: G440)

Cycle run

Touch Probe Cycle 440 measures the axis shifts of the machine. Make sure that the cylindrical calibrating tool used in connection with the TT 130 has the correct dimensions.

1. The TNC positions the calibrating tool at rapid traverse (value from MP6550) and following the positioning logic (refer to section 1.2) in the vicinity of the TT.
2. At first the TNC makes a measurement in the touch probe axis. The calibrating tool is offset by the value you have defined in the tool table TOOL.T under TT: R-OFFS (standard = tool radius). The TNC always performs the measurement in the touch probe axis.
3. Then the TNC makes the measurement in the working plane. You define via parameter Q364 in which axis and in which direction of the working plane the measurement is to be made.
4. If you make a calibration, the TNC saves the calibration data. Whenever you make a measurement, the TNC compares the measured values to the calibration data and writes the deviations to the following Q parameters:

Parameter number	Meaning
Q185	Deviation from calibration value in X
Q186	Deviation from calibration value in Y
Q187	Deviation from calibration value in Z

   You can use this value for compensating the deviation through an incremental datum shift (Cycle 7).
5. Finally, the calibrating tool returns to the clearance height.

Please note while programming:

Before running Cycle 440 for the first time, you must have calibrated the TT tool touch probe with the TT Cycle 30.

Ensure that the tool data of the calibrating tool has been entered in the tool table TOOL.T.

Before running the cycle, you must activate the calibrating tool with TOOL CALL.

Ensure that the TT tool touch probe is connected to input X13 of the logic unit and is ready to function (MP65xx).

Before you perform a measurement, you must have made at least one calibration, otherwise the TNC will output an error message. If you are working with several traverse ranges, you have to make a calibration for each of them.

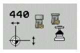
The TNC calculates incorrect values if the probing directions for calibrating and measuring do not correspond.

Each time you run Cycle 440, the TNC resets the result parameters Q185 to Q187.

If you want to set a limit for the axis shift in the machine axes, enter the desired limits in the tool table TOOL.T under LTOL for the spindle axis and under RTOL for the working plane. If the limits are exceeded, the TNC outputs a corresponding error message after a verification measurement.

After the cycle is completed, the TNC restores the spindle settings that were active before the cycle (**M3/M4**).

Cycle parameters



- **Operation: 0=calibr., 1=measure?** Q363: Specify whether you want to calibrate or make a verification measurement:
  0: Calibrate
  1: Measure
- **Probing directions** Q364: Definition of probing direction(s) in the working plane:
  0: Measuring only in the positive direction of the reference axis
  1: Measuring only in the positive direction of the minor axis
  2: Measuring only in the negative direction of the reference axis
  3: Measuring only in the negative direction of the minor axis
  4: Measuring in the positive directions of the reference axis and the minor axis
  5: Measuring in the positive direction of the reference axis and in the negative direction of the minor axis
  6: Measuring in the negative direction of the reference axis and in the positive direction of the minor axis
  7: Measuring in the negative directions of the reference axis and the minor axis
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and probe contact. Q320 is added to MP6540. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur (referenced to the active datum). Input range -99999.9999 to 99999.9999; alternatively **PREDEF**

Example: NC blocks

```
5 TCH PROBE 440 MEASURE AXIS SHIFT
    Q363=1    ;DIRECTION
    Q364=0    ;PROBING DIRECTIONS
    Q320=2    ;SET-UP CLEARANCE
    Q260=+50  ;CLEARANCE HEIGHT
```

17.7 FAST PROBING (Cycle 441, DIN/ISO: G441, FCL 2 function)

Cycle run

Touch Probe Cycle 441 allows the global setting of different touch probe parameters (e.g. positioning feed rate) for all subsequently used touch probe cycles. This makes it easy to optimize the programs so that reductions in total machining time are achieved.

Please note while programming:

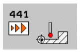
Before programming, note the following

There are no machine movements contained in Cycle 441. It only sets different probing parameters.

END PGM, **M02**, **M30** resets the global settings of Cycle 441.

You can activate automatic angle tracking (Cycle Parameter **Q399**) only if Machine Parameter 6165=1. If you change MP6165, you must recalibrate the touch probe.

Cycle parameters



- **Positioning feed rate** Q396: Define the feed rate at which the touch probe is moved to the specified positions. Input range 0 to 99999.9999
- **Positioning feed rate=FMAX (0/1)** Q397: Define whether the touch probe is to move at **FMAX** (rapid traverse) to the specified positions:
  0: Move at feed rate from **Q396**
  1: Move at **FMAX**
  If your machine provides separate potentiometers for rapid traverse and feed rate, the feed rate can only be controlled with the feed-rate potentiometer even if you enter Q397=1.
- **Angle tracking** Q399: Define whether the TNC is to orient the touch probe before each probing process:
  0: Do not orient
  1: Orient the spindle before each probing process to increase the accuracy
- **Automatic interruption** Q400: Define whether the TNC is to interrupt program run and display the measurement results on the screen after a measuring cycle for automatic workpiece measurement:
  0: Never interrupt the program run, not even if the output of the measurement results on the screen is selected in the respective probing cycle.
  1: Always interrupt program run and display the measurement results on the screen. To continue the program run, press the NC Start button.

Example: NC blocks

```
5 TCH PROBE 441 FAST PROBING
    Q396=3000 ;POSITIONING FEED RATE
    Q397=0    ;SELECT FEED RATE
    Q399=1    ;ANGLE TRACKING
    Q400=1    ;INTERRUPTION
```

17.8 CALIBRATE TS (Cycle 460, DIN/ISO: G460)

Cycle run

With Cycle 460 you can calibrate a triggering 3-D touch probe automatically on an exact calibration sphere. You can do radius calibration alone, or radius and length calibration.

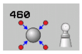
1. Clamp the calibration sphere and check for potential collisions.
2. In the touch probe axis, position the touch probe over the calibration sphere, and in the working plane, approximately over the sphere center.
3. The first movement in the cycle is in the negative direction of the touch probe axis.
4. Then the cycle determines the exact center of the sphere in the touch probe axis.

Please note while programming:

Before programming, note the following

Pre-position the touch probe in the program so that it is located approximately above the center of the calibration sphere.

Cycle parameters



- **Exact calibration sphere radius** Q407: Enter the exact radius of the calibration sphere used. Input range 0.0001 to 99.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
  Alternative: **PREDEF**
- **No. of probe points in plane (4/3)** Q423: Specify whether the TNC should measure the calibration sphere in the plane with 4 or 3 probing points. 3 probing points increase the measuring speed:
  4: Use 4 measuring points (default setting)
  3: Use 3 measuring points
- **Reference angle** Q380 (absolute): Reference angle (basic rotation) for measuring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. Input range 0 to 360.0000
- **Calibrate length (0/1)** Q433: Define whether the TNC should also calibrate the touch probe length after radius calibration:
  0: Do not calibrate touch probe length
  1: Calibrate touch probe length
- **Datum for length** Q434 (absolute): Coordinate of the calibration sphere center. The definition is only required if length calibration is to be carried out. Input range -99999.9999 to 99999.9999

Example: NC blocks

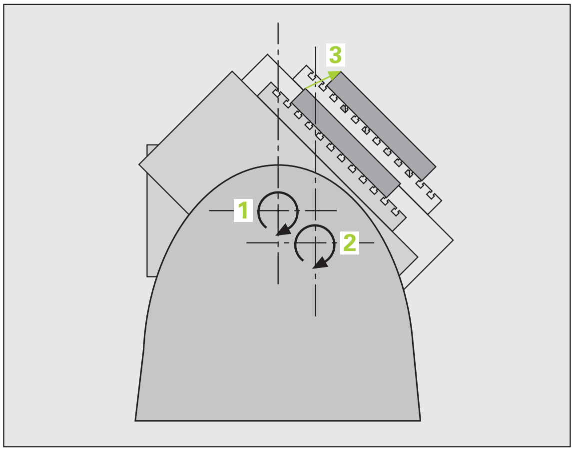
```
5 TCH PROBE 460 CALIBRATE TS
    Q407=12.5 ;SPHERE RADIUS
    Q320=0    ;SET-UP CLEARANCE
    Q301=1    ;MOVE TO CLEARANCE
    Q423=4    ;NO. OF PROBE POINTS
    Q380=+0   ;REFERENCE ANGLE
    Q433=0    ;CALIBRATE LENGTH
    Q434=-2.5 ;DATUM
```

18

Touch Probe Cycles: Automatic Kinematics Measurement

18.1 Kinematics Measurement with TS Touch Probes (KinematicsOpt Option)

Fundamentals



Accuracy requirements are becoming increasingly stringent, particularly in the area of 5-axis machining. Complex parts need to be manufactured with precision and reproducible accuracy even over long periods.

Some of the reasons for inaccuracy in multi-axis machining are deviations between the kinematic model saved in the control (see **1** in the figure at right), and the kinematic conditions actually existing on the machine (see **2** in the figure at right). When the rotary axes are positioned, these deviations cause inaccuracy of the workpiece (see **3** in the figure at right). It is therefore necessary for the model to approach reality as closely as possible.

The new TNC function **KinematicsOpt** is an important component that helps you to really fulfill these complex requirements: A 3-D touch probe cycle measures the rotary axes on your machine fully automatically, regardless of whether they are in the form of tables or spindle heads. A calibration sphere is fixed at any position on the machine table, and measured with a resolution that you define. During cycle definition you simply define for each rotary axis the area that you want to measure.

From the measured values, the TNC calculates the static tilting accuracy. The software minimizes the positioning error arising from the tilting movements and, at the end of the measurement process, automatically saves the machine geometry in the respective machine constants of the kinematic table.

Overview

The TNC offers cycles that enable you to automatically save, check and optimize the machine kinematics:

Cycle	Soft key	Page
450 SAVE KINEMATICS: Automatically saving and restoring kinematic configurations	450	Page 482
451 MEASURE KINEMATICS: Automatically checking or optimizing the machine kinematics	451	Page 484
452 PRESET COMPENSATION: Automatically checking or optimizing the machine kinematics	452	Page 500

18.2 Prerequisites

The following are prerequisites for using the KinematicsOpt option:

- The software options 48 (KinematicsOpt) and 8 (software option 1) and FCL3 must be enabled.
- Software option 52 (KinematicsComp) is necessary for compensations of angular positions.
- The 3-D touch probe used for the measurement must be calibrated.
- The cycles can only be carried out with the tool axis Z.
- A calibration sphere with an exactly known radius and sufficient rigidity must be attached to any position on the machine table. HEIDENHAIN recommends using the calibration spheres **KKH 250** (part number 655 475-01) or **KKH 100** (part number 655 475-02), which have particularly high rigidity and are designed especially for machine calibration. Please contact HEIDENHAIN if you have any questions in this regard.
- The kinematics description of the machine must be complete and correct. The transformation values must be entered with an accuracy of approx. 1 mm.
- The complete machine geometry must have been measured (by the machine tool builder during commissioning).
- **MP6600** must define the tolerance limit starting from which the TNC displays a note if the changes in the kinematic data exceed this limit value (see "KinematicsOpt: Tolerance limit in Optimization mode: MP6600" on page 335).
- **MP6601** must define the maximum permissible deviation from the entered cycle parameter by the calibration sphere radius measured in the cycles (see "KinematicsOpt, permissible deviation of the calibration ball radius: MP6601" on page 335).
- The M function number to be used for rotary axis positioning must be entered in **MP6602**, or –1 if positioning is to be done by the NC. An M function must be specially provided for this application by your machine tool builder.

Please note while programming:

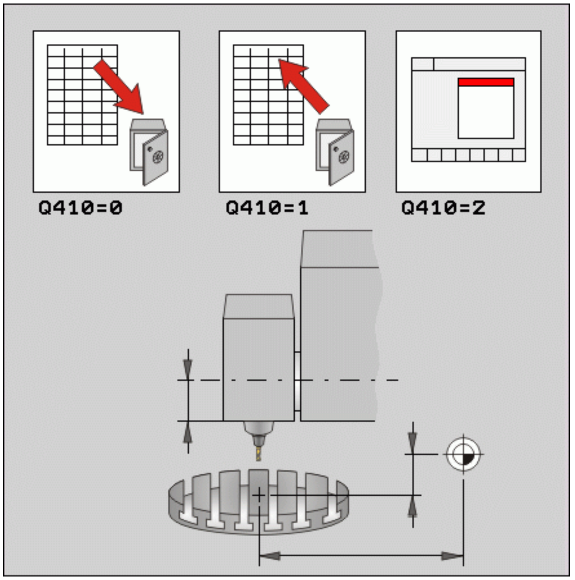
The KinematicsOpt cycles use the global string parameters **QS0** to **QS99**. Please note that they may have changed after execution of these cycles.

If MP6602 is not equal to –1, you have to position the rotary axes to 0 degrees (ACTUAL system) before starting one of the KinematicsOpt cycles (except for Cycle 450).

18.3 SAVE KINEMATICS (Cycle 450, DIN/ISO: G450; Option)

Cycle run

With touch probe cycle 450, you can save the active machine kinematics, restore a previously saved one, or output the current saving status on the screen and in a log file. There are 10 memory spaces available (numbers 0 to 9).



Please note while programming:

Always save the active kinematics configuration before running a kinematics optimization. Advantage:

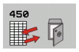
- You can restore the old data if you are not satisfied with the results or if errors occur during optimization (e.g. power failure).

Save mode: In addition to the kinematic configuration, the TNC always saves the code number (freely definable) last entered under MOD. Then you cannot overwrite this memory space unless you enter this code number. If you have saved a kinematic configuration without a code number, the TNC automatically overwrites this memory space during the next saving process!

Restore mode: The TNC can restore saved data only to a matching kinematic configuration.

Restore mode: Note that a change in the kinematics always changes the preset as well. Set the preset again if necessary.

Cycle parameters



- **Mode (0/1/2)** Q410: Specify whether to save or restore a kinematics configuration:
  0: Save active kinematics
  1: Restore previously saved kinematics configuration
  2: Display the storage status
- **Memory (0...9)** Q409: Number of the memory space to which you want to save the entire kinematics configuration, or the number of the memory space from which you want to restore it. Input range 0 to 9, no function if mode 2 is selected.

Example: NC blocks

```
5 TCH PROBE 450 SAVE KINEMATICS
    Q410=0    ;MODE
    Q409=1    ;MEMORY
```

Log function

After running Cycle 450, the TNC creates a measuring log (**TCHPR450.TXT**) containing the following information:

- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Mode used (0=Save/1=Restore/2=Saving status)?
- Number of the memory space (0 to 9)
- Line number of the kinematics configuration in the kinematic table
- Code number, if you entered one immediately before running Cycle 450

The other data in the log vary depending on the selected mode:

- Mode 0:
  Logging of all axis entries and transformation entries of the kinematics chain that the TNC has saved.
- Mode 1:
  Logging of all transformation entries before and after restoring the kinematics configuration
- Mode 2:
  List with the current saving status on the screen and in the log, including the number of the memory space, code numbers, kinematics numbers and date of saving

18.4 MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451; Option)

Cycle run

The touch probe cycle 451 enables you to check and, if required, optimize the kinematics of your machine. Use the 3-D TS touch probe to measure a HEIDENHAIN calibration sphere that you have attached to the machine table.

HEIDENHAIN recommends using the calibration spheres **KKH 250** (part number 655 475-01) or **KKH 100** (part number 655 475-02), which have particularly high rigidity and are designed especially for machine calibration. Please contact HEIDENHAIN if you have any questions in this regard.

The TNC evaluates the static tilting accuracy. The software minimizes the spatial error arising from the tilting movements and, at the end of the measurement process, automatically saves the machine geometry in the respective machine constants of the kinematics description.

1. Clamp the calibration sphere and check for potential collisions.
2. In the Manual Operation mode, set the datum in the center of the sphere, or if **Q431=1** or **Q431=3** is defined: In the touch probe axis, manually position the touch probe over the calibration sphere , and in the working plane, over the sphere center.
3. Select the Program Run mode and start the calibration program.

4 The TNC automatically measures all the rotary axes successively in the resolution you defined. The current measurement status is displayed in a pop-up window. The TNC hides the status window when the distance to be traversed is greater than the radius of the ball tip.

5 The TNC saves the measured values in the following Q parameters:

Parameter number	Meaning
Q141	Standard deviation measured in the A axis (-1 if axis was not measured)
Q142	Standard deviation measured in the B axis (-1 if axis was not measured)
Q143	Standard deviation measured in the C axis (-1 if axis was not measured)
Q144	Optimized standard deviation in the A axis (–1 if axis was not optimized)
Q145	Optimized standard deviation in the B axis (–1 if axis was not optimized)
Q146	Optimized standard deviation in the C axis (–1 if axis was not optimized)
Q147	Offset error in X direction, for manual transfer to the corresponding machine parameter
Q148	Offset error in Y direction, for manual transfer to the corresponding machine parameter
Q149	Offset error in Z direction, for manual transfer to the corresponding machine parameter

Positioning direction

The positioning direction of the rotary axis to be measured is determined from the start angle and the end angle that you define in the cycle. A reference measurement is automatically performed at 0°. The TNC will issue an error message if the selected start angle, end angle and number of measuring points result in a measuring position of 0°.

Specify the start and end angles to ensure that the same position is not measured twice. As mentioned above, a duplicated point measurement (e.g. measuring positions +90° and -270°) is not advisable, however it does not cause an error message.

- Example: Start angle = +90°, end angle = –90°
  - Start angle = +90°
  - End angle = -90°
  - No. of measuring points = 4
  - Stepping angle resulting from the calculation = (-90 - +90) / (4-1) = –60°
  - Measuring point 1= +90°
  - Measuring point 2= +30°
  - Measuring point 3= -30°
  - Measuring point 4= -90°
- Example: Start angle = +90°, end angle = +270°
  - Start angle = +90°
  - End angle = +270°
  - No. of measuring points = 4
  - Stepping angle resulting from the calculation = (270 – 90) / (4 – 1) = +60°
  - Measuring point 1= +90°
  - Measuring point 2= +150°
  - Measuring point 3= +210°
  - Measuring point 4= +270°

Machines with Hirth-coupled axes

Danger of collision!

In order to be positioned, the axis must move out of the Hirth coupling. So remember to leave a large enough safety clearance to prevent any risk of collision between the touch probe and calibration sphere. Also ensure that there is enough space to reach the safety clearance (software limit switch).

Define a retraction height **Q408** greater than 0 if software option 2 (**M128**, **FUNCTION TCPM**) is not available.

If necessary, the TNC rounds the calculated measuring positions so that they fit into the Hirth grid (depending on the start angle, end angle and number of measuring points).

Depending on the machine configuration, the TNC cannot position the rotary axes automatically. If this is the case, you need a special M function from the machine tool builder enabling the TNC to move the rotary axes. The machine manufacturer must have entered the number of the M function in **MP6602** for this purpose.

The measuring positions are calculated from the start angle, end angle and number of measurements for the respective axis and from the Hirth grid.

Example calculation of measuring positions for an A axis:

Start angle **Q411** = -30

End angle **Q412** = +90

Number of measuring points **Q414** = 4

Hirth grid = 3°

Calculated stepping angle = (Q412 – Q411) / (Q414 – 1)

Calculated stepping angle = = (90 – –30) / (4 – 1) = 120 / 3 = 40

Measuring position 1 = Q411 + 0 * stepping angle = -30° --> -30°

Measuring position 2 = Q411 + 1 * stepping angle = +10° --> 9°

Measuring position 3 = Q411 + 2 * stepping angle = +50° --> 51°

Measuring position 4 = Q411 + 3 * stepping angle = +90° --> 90°

Choice of number of measuring points

To save time, you can make a rough optimization with a small number of measuring points (1-2).

You then make a fine optimization with a medium number of measuring points (recommended value = 4). Higher numbers of measuring points do not usually improve the results. Ideally, you should distribute the measuring points evenly over the tilting range of the axis.

This is why you should measure an axis with a tilting range of 0° to 360° at three measuring points, namely at 90°, 180° and 270°.

If you want to check the accuracy accordingly, you can enter a higher number of measuring points in the **Check** mode.

You must not define a measuring point at 0° or 360°. These positions do not provide any metrologically relevant data and lead to an error message!

Choice of the calibration sphere position on the machine table

In principle, you can fix the calibration sphere to any accessible position on the machine table and also on fixtures or workpieces. The following factors can positively influence the result of measurement:

- On machines with rotary tables/tilting tables:
  Clamp the calibration sphere as far as possible away from the center of rotation.
- Machines with large traverse:
  Clamp the calibration sphere as closely as possible to the position intended for subsequent machining.

Notes on the accuracy

The geometrical and positioning error of the machine influences the measured values and therefore also the optimization of a rotary axis. For this reason there will always be a certain amount of error.

If there were no geometrical and positioning errors, any values measured by the cycle at any point on the machine at a certain time would be exactly reproducible. The greater the geometrical and positioning error, the greater is the dispersion of measured results when you fix the calibration sphere to different positions in the machine coordinate system.

The dispersion of results recorded by the TNC in the measuring log is a measure of the machine's static tilting accuracy. However, the measuring circle radius and the number and position of measuring points have to be included in the evaluation of accuracy. One measuring point alone is not enough to calculate dispersion. For only one point, the result of the calculation is the spatial error of that measuring point.

If several rotary axes are moved simultaneously, their error values are combined. In the worst case they are added together.

If your machine is equipped with a controlled spindle, you should activate the angle tracking using **MP6165.** This generally increases the accuracy of measurements with a 3-D touch probe.

If required, deactivate the lock on the rotary axes for the duration of the calibration. Otherwise it may falsify the results of measurement. The machine manual provides further information.

Notes on various calibration methods

- **Rough optimization during commissioning after entering approximate dimensions.**
  - Number of measuring points between 1 and 2
  - Angular step of the rotary axes: Approx. 90°
- **Fine optimization over the entire range of traverse**
  - Number of measuring points between 3 and 6
  - The start and end angles should cover the largest possible traverse range of the rotary axes
  - Position the calibration sphere on the machine table so that on rotary table axes there is a large measuring circle, or so that on swivel head axes the measurement can be made at a representative position (e.g. in the center of the traverse range).
- **Optimization of a specific rotary axis position**
  - Number of measuring points between 2 and 3
  - The measurements are made near the rotary axis angle at which the workpiece is to be machined
  - Position the calibration sphere on the machine table for calibration at the position subsequently intended for machining
- **Inspecting the machine accuracy**
  - Number of measuring points between 4 and 8
  - The start and end angles should cover the largest possible traverse range of the rotary axes
- **Determination of the rotary axis backlash**
  - Number of measuring points between 8 and 12
  - The start and end angles should cover the largest possible traverse range of the rotary axes

Backlash

Backlash is a small amount of play between the rotary or angle encoder and the table that occurs when the traverse direction is reversed. If the rotary axes have backlash outside of the control loop, for example because the angle measurement is made with the motor encoder, this can result in significant error during tilting.

With input parameter **Q432** you can activate backlash measurement. Enter an angle that the TNC uses as traversing angle. The cycle will then carry out two measurements per rotary axis. If you take over the angle value 0, the TNC will not measure any backlash.

The TNC does not perform an automatic backlash compensation.

If the measuring circle radius is < 1 mm, the TNC does not calculate the backlash. The larger the measuring circle radius, the more accurately the TNC can determine the rotary axis backlash (see also "Log function" on page 497).

Backlash measurement is not possible if **MP6602** is set or if the axis is a Hirth axis.

Please note while programming:

Note that all functions for tilting in the working plane are reset. **M128** and **FUNCTION TCPM** are deactivated.

Position the calibration sphere on the machine table so that there can be no collisions during the measuring process.

Before defining the cycle you must set the datum in the center of the calibration sphere and activate it, or you define the input parameter Q431 correspondingly to 1 or 3.

If **MP6602** is not equal to –1 (PLC macro positions the rotary axes), only start a measurement if all rotary axes are at 0°.

For the positioning feed rate when moving to the probing height in the touch probe axis, the TNC uses the value from cycle parameter **Q253** or **MP6150**, whichever is smaller. The TNC always moves the rotary axes at positioning feed rate **Q253**, while the probe monitoring is inactive.

If the kinematic data attained in the optimize mode are greater than the permissible limit (**MP6600**), the TNC shows a warning. Then you have to confirm acceptance of the attained value by pressing NC start.

Note that a change in the kinematics always changes the preset as well. After an optimization, reset the preset.

In every probing process the TNC first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than you have defined in **MP6601,** the TNC shows an error message and ends the measurement.

If you interrupt the cycle during the measurement, the kinematic data might no longer be in the original condition. Save the active kinematic configuration before an optimization with Cycle 450, so that in case of a failure the most recently active kinematic configuration can be restored.

Programming in inches: The TNC always records the log data and results of measurement in millimeters.

The TNC ignores cycle definition data that applies to inactive axes.

Cycle parameters

- **Mode (0/1/2)** Q406: Specify whether the TNC should check or optimize the active kinematics:
  0: Check the active machine kinematics. The TNC measures the kinematics in the rotary axes you have defined, but it does not make any changes to it. The TNC displays the results of measurement in a measurement log
  1: Optimize the active machine kinematics. The TNC measures the kinematics in the rotary axes you have defined and **optimizes the position** of the rotary axes of the active kinematics.
  2: Optimize the active machine kinematics. The TNC measures the kinematics in the rotary axes you have defined and **optimizes the position and compensates the angle** of the rotary axes of the active kinematics. The KinematicsComp option must be enabled for Mode 2.
- **Exact calibration sphere radius** Q407: Enter the exact radius of the calibration sphere used. Input range 0.0001 to 99.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Retraction height** Q408 (absolute): Input range 0.0001 to 99999.9999
  - Input 0:
    Do not move to any retraction height. The TNC moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The TNC moves to the first measuring position in the sequence A, then B, then C.
  - Input >0:
    Retraction height in the untilted workpiece coordinate system to which the TNC positions before a rotary axis positioning in the spindle axis. Also, the TNC moves the touch probe in the working plane to the datum. Probe monitoring is not active in this mode. Define the positioning velocity in parameter Q253.

Example: Calibration program

```
4 TOOL CALL "TCH PROBE" Z
5 TCH PROBE 450 SAVE KINEMATICS
     Q410=0     ;MODE
     Q409=5     ;MEMORY
6 TCH PROBE 451 MEASURE KINEMATICS
     Q406=1     ;MODE
     Q407=12.5 ;SPHERE RADIUS
     Q320=0     ;SET-UP CLEARANCE
     Q408=0     ;RETR. HEIGHT
     Q253=750   ;F PRE-POSITIONING
     Q380=0     ;REFERENCE ANGLE
     Q411=-90   ;START ANGLE A AXIS
     Q412=+90   ;END ANGLE A AXIS
     Q413=0     ;INCID. ANGLE A AXIS
     Q414=0     ;MEAS. POINTS A AXIS
     Q415=-90   ;START ANGLE B AXIS
     Q416=+90   ;END ANGLE B AXIS
     Q417=0     ;INCID. ANGLE B AXIS
     Q418=2     ;MEAS. POINTS B AXIS
     Q419=-90   ;START ANGLE C AXIS
     Q420=+90   ;END ANGLE C AXIS
     Q421=0     ;INCID. ANGLE C AXIS
     Q422=2     ;MEAS. POINTS C AXIS
     Q423=4     ;NO. OF MEAS. POINTS
     Q431=1     ;PRESET
     Q432=0     ;BACKLASH, ANG. RANGE
```

- **Feed rate for pre-positioning** Q253: Traversing speed of the tool in mm/min during positioning. Input range 0.0001 to 99999.9999; alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Reference angle** Q380 (absolute): Reference angle (basic rotation) for measuring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. Input range 0 to 360.0000
- **Start angle A axis** Q411 (absolute): Starting angle in the A axis at which the first measurement is to be made. Input range -359.999 to 359.999
- **End angle A axis** Q412 (absolute): Ending angle in the A axis at which the last measurement is to be made. Input range -359.999 to 359.999
- **Angle of incid. A axis** Q413: Angle of incidence in the A axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999
- **Number meas. points A axis** Q414: Number of probe measurements with which the TNC is to measure the A axis. If the input value = 0, the TNC does not measure the respective axis. Input range 0 to 12
- **Start angle B axis** Q415 (absolute): Starting angle in the B axis at which the first measurement is to be made. Input range -359.999 to 359.999
- **End angle B axis** Q416 (absolute): Ending angle in the B axis at which the last measurement is to be made. Input range -359.999 to 359.999
- **Angle of incid. in B axis** Q417: Angle of incidence in the B axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999
- **Number meas. points B axis** Q418: Number of probe measurements with which the TNC is to measure the B axis. If the input value = 0, the TNC does not measure the respective axis. Input range 0 to 12

- **Start angle C axis** Q419 (absolute): Starting angle in the C axis at which the first measurement is to be made. Input range -359.999 to 359.999
- **End angle C axis** Q420 (absolute): Ending angle in the C axis at which the last measurement is to be made. Input range -359.999 to 359.999
- **Angle of incid. in C axis** Q421: Angle of incidence in the C axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999
- **Number meas. points C axis** Q422: Number of probe measurements with which the TNC is to measure the C axis. Input range 0 to 12. If the input value = 0, the TNC does not measure the respective axis
- **No. of measuring points** Q423: Specify the number of probing points to be used by the TNC for measuring the calibration sphere in the plane. Input range: 3 to 8 measurements
- **Preset (0/1/2/3)** Q431: Specify whether the TNC is to set the active preset (datum) automatically in the center of the sphere:
  0: Do not set the preset automatically in the center of the sphere: Set the preset manually before the start of the cycle
  1: Set the preset automatically in the center of the sphere before measurement: Pre-position the touch probe manually over the calibration sphere before the start of the cycle
  2: Set the preset automatically in the center of the sphere after measurement: Set the preset manually before the start of the cycle
  3: Set the preset in the center of the sphere before and after measurement: Pre-position the touch probe manually over the calibration sphere before the start of the cycle
- **Backlash, angle range** Q432: Here you define the angle value to be used as traverse for the measurement of the rotary axis backlash. The traversing angle must be significantly larger than the actual backlash of the rotary axes. If input value = 0, the TNC does not measure the backlash. Input range -3.0000 to +3.0000

If you have activated "Preset" before measurement (Q431 = 1/3), then move the touch probe to a position above the center of the calibration sphere before the start of the cycle.

Various modes (Q406)

- **Test mode Q406 = 0**
  - The TNC measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
  - The TNC records the results of a possible position optimization but does not make any adjustments.
- **Position Optimization mode Q406 = 1**
  - The TNC measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
  - During this, the TNC tries to change the position of the rotary axis in the kinematics model in order to achieve higher accuracy.
  - The machine data is adjusted automatically.
- **Position and Angle Optimization mode Q406 = 2**
  - The TNC measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
  - First the TNC tries to optimize the angular orientation of the rotary axis by means of compensation (Option #52, KinematicsComp).
  - If the TNC succeeded in optimizing the angular orientation, it then optimizes the position through another measurement series.

For angle optimization, the machine manufacturer must have adapted the configuration correspondingly. You can ask your machine manufacturer if this is the case, and whether an angle optimization makes sense. Angle optimization can be particularly useful on small, compact machines.

Angle compensation is only possible with Option #52 **KinematicsComp**.

Example: Angle and position optimization of the rotary axes after automatic datum setting

```
1 TOOL CALL "TS640" Z
2 TCH PROBE 451 MEASURE KINEMATICS
     Q406=2    ;MODE
     Q407=12.5 ;SPHERE RADIUS
     Q320=0    ;SET-UP CLEARANCE
     Q408=0    ;RETR. HEIGHT
     Q253=750  ;F PRE-POSITIONING
     Q380=0    ;REFERENCE ANGLE
     Q411=-90  ;START ANGLE A AXIS
     Q412=+90  ;END ANGLE A AXIS
     Q413=0    ;INCID. ANGLE A AXIS
     Q414=0    ;MEAS. POINTS A AXIS
     Q415=-90  ;START ANGLE B AXIS
     Q416=+90  ;END ANGLE B AXIS
     Q417=0    ;INCID. ANGLE B AXIS
     Q418=4    ;MEAS. POINTS B AXIS
     Q419=+90  ;START ANGLE C AXIS
     Q420=+270 ;END ANGLE C AXIS
     Q421=0    ;INCID. ANGLE C AXIS
     Q422=3    ;MEAS. POINTS C AXIS
     Q423=3    ;NO. OF MEAS. POINTS
     Q431=1    ;PRESET
     Q432=0    ;BACKLASH, ANG. RANGE
```

Log function

After running Cycle 451, the TNC creates a measuring log **(TCHPR451.TXT)** containing the following information:

- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Mode used (0=Check/1=Optimize position/2=Optimize pose)
- Active kinematic number
- Entered calibration sphere radius
- For each measured rotary axis:
  - Starting angle
  - End angle
  - Angle of incidence
  - Number of measuring points
  - Dispersion (standard deviation)
  - Maximum error
  - Angular error
  - Averaged backlash
  - Averaged positioning error
  - Measuring circle radius
  - Compensation values in all axes (preset shift)
  - Evaluation of measuring points
  - Measurement uncertainty of rotary axes

Notes on log data

- **Error outputs**
  In the Test mode (**Q406=0**) the TNC outputs the accuracy that can be attained by optimization and/or the accuracies attained through optimization (Modes 1 and 2).
  If the angular position of a rotary axis was calculated, the measured data is also shown in the log.
- **Dispersion (standard deviation)**
  In the log, 'dispersion', a term from statistics, is used as a measure of accuracy. **Measured dispersion** (measured standard deviation) means that 68.3 % of the actually measured spatial errors are within the specified range (+/–). **Optimized dispersion** (optimized standard deviation) means that 68.3% of the spatial errors to be expected after the correction of the kinematics are within the specified range (+/–).
- **Evaluation of measuring points**
  The valuation numbers are a measure of the quality of the measuring positions with respect to the changeable transformations of the kinematics model. The higher the valuation number, the greater the benefit from optimization by the TNC. The valuation of any rotary axis should not fall below a value of **2.** Values greater than or equal to **4** are desirable If the valuation numbers are too small, increase the measurement range of the rotary axis, or the number of measuring points.

If the valuation numbers are too small, increase the measurement range of the rotary axis, or the number of measuring points. If these measures do not improve the valuation number, this might be due to an incorrect kinematics description. If necessary, inform your service agency.

Measurement uncertainty of angles

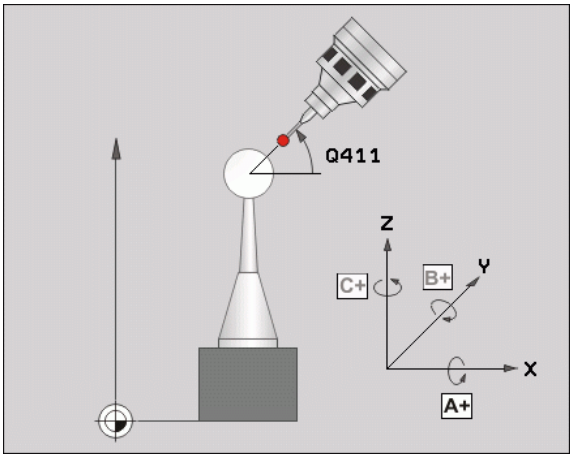
The TNC always indicates measurement uncertainty in degrees per 1 µm of system uncertainty. This information is important for evaluating the quality of the measured positioning errors, or the backlash of a rotary axis.

The system uncertainty includes at least the repeatability of the axes (backlash) as well as the positioning uncertainty of the linear axes (positioning errors) and of the touch probe. Since the TNC does not know the accuracy of the complete system, you must make a separate evaluation.

- Example of uncertainty of the calculated positioning errors:
  - Positioning uncertainty of each linear axis: 10 µm
  - Uncertainty of touch probe: 2 µm
  - Logged measurement uncertainty: 0.0002 °/µm
  - System uncertainty = SQRT(3 * 10^2 + 2^2) = 17.4 µm
  - Measurement uncertainty = 0.0002 °/µm * 17.4 µm = 0.0034°
- Example of uncertainty of the calculated backlash:
  - Repeatability of each linear axis: 5 µm
  - Uncertainty of touch probe: 2 µm
  - Logged measurement uncertainty: 0.0002 °/µm
  - System uncertainty = SQRT(3 * 5^2 + 2^2) = 8.9 µm
  - Measurement uncertainty = 0.0002 °/µm * 8.9 µm = 0.0018°

18.5 PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, Option)

Cycle run



Touch probe cycle 452 optimizes the kinematic transformation chain of your machine (see "MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451; Option)" on page 484). Then the TNC corrects the workpiece coordinate system in the kinematics model in such a way that the current preset is in the center of the calibration sphere after optimization.

This cycle enables you, for example, to adjust different interchangeable heads so that the workpiece preset applies for all heads.

1. Clamp the calibration sphere.
2. Measure the complete reference head with Cycle 451, and use Cycle 451 to finally set the preset in the center of the sphere.
3. Insert the second head.
4. Use Cycle 452 to measure the interchangeable head up to the point where the head is changed.
5. Use Cycle 452 to adjust other interchangeable heads to the reference head.

If it is possible to leave the calibration sphere clamped to the machine table during machining, you can compensate for machine drift, for example. This procedure is also possible on a machine without rotary axes.

1. Clamp the calibration sphere and check for potential collisions.
2. Set the preset in the calibration sphere.
3. Set the preset on the workpiece, and start machining the workpiece.
4. The TNC automatically measures all the rotary axes successively in the resolution you defined. The current measurement status is displayed in a pop-up window. The TNC hides the status window when the distance to be traversed is greater than the radius of the ball tip.
5. Use Cycle 452 for preset compensation at regular intervals. The TNC measures the drift of the axes involved and compensates it in the kinematics description.

Parameter number	Meaning
Q141	Standard deviation measured in the A axis (-1 if axis was not measured)
Q142	Standard deviation measured in the B axis (-1 if axis was not measured)
Q143	Standard deviation measured in the C axis (-1 if axis was not measured)
Q144	Optimized standard deviation in the A axis (–1 if axis was not measured)
Q145	Optimized standard deviation in the B axis (–1 if axis was not measured)
Q146	Optimized standard deviation in the C axis (–1 if axis was not measured)
Q147	Offset error in X direction, for manual transfer to the corresponding machine parameter
Q148	Offset error in Y direction, for manual transfer to the corresponding machine parameter
Q149	Offset error in Z direction, for manual transfer to the corresponding machine parameter

Please note while programming:

In order to be able to perform a preset compensation, the kinematics must be specially prepared. The machine manual provides further information.

Note that all functions for tilting in the working plane are reset. **M128** and **FUNCTION TCPM** are deactivated.

Position the calibration sphere on the machine table so that there can be no collisions during the measuring process.

Before defining the cycle you must set the datum in the center of the calibration sphere and activate it.

For rotary axes without separate position encoders, select the measuring points in such a way that you have to traverse a distance of 1° to the limit switch. The TNC needs this distance for internal backlash compensation.

For the positioning feed rate when moving to the probing height in the touch probe axis, the TNC uses the value from cycle parameter **Q253** or MP6150, whichever is smaller. The TNC always moves the rotary axes at positioning feed rate **Q253**, while the probe monitoring is inactive.

If the kinematic data attained in the optimize mode are greater than the permissible limit (**MP6600**), the TNC shows a warning. Then you have to confirm acceptance of the attained value by pressing NC start.

Note that a change in the kinematics always changes the preset as well. After an optimization, reset the preset.

In every probing process the TNC first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than you have defined in **MP6601,** the TNC shows an error message and ends the measurement.

If you interrupt the cycle during the measurement, the kinematic data might no longer be in the original condition. Save the active kinematic configuration before an optimization with Cycle 450, so that in case of a failure the most recently active kinematic configuration can be restored.

Programming in inches: The TNC always records the log data and results of measurement in millimeters.

Cycle parameters

- **Exact calibration sphere radius** Q407: Enter the exact radius of the calibration sphere used. Input range 0.0001 to 99.9999
- **Set-up clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140. Input range 0 to 99999.9999; alternatively **PREDEF**
- **Retraction height** Q408 (absolute): Input range 0.0001 to 99999.9999
  - Input 0:
    Do not move to any retraction height. The TNC moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The TNC moves to the first measuring position in the sequence A, then B, then C.
  - Input >0:
    Retraction height in the untilted workpiece coordinate system to which the TNC positions before a rotary axis positioning in the spindle axis. Also, the TNC moves the touch probe in the working plane to the datum. Probe monitoring is not active in this mode. Define the positioning velocity in parameter Q253.
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool in mm/min during positioning. Input range 0.0001 to 99999.9999; alternatively **FMAX**, **FAUTO** **PREDEF**
- **Reference angle** Q380 (absolute): Reference angle (basic rotation) for measuring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. Input range 0 to 360.0000
- **Start angle A axis** Q411 (absolute): Starting angle in the A axis at which the first measurement is to be made. Input range -359.999 to 359.999
- **End angle A axis** Q412 (absolute): Ending angle in the A axis at which the last measurement is to be made. Input range -359.999 to 359.999
- **Angle of incid. A axis** Q413: Angle of incidence in the A axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999
- **Number meas. points A axis** Q414: Number of probe measurements with which the TNC is to measure the A axis. If the input value = 0, the TNC does not measure the respective axis. Input range 0 to 12
- **Start angle B axis** Q415 (absolute): Starting angle in the B axis at which the first measurement is to be made. Input range -359.999 to 359.999

Example: Calibration program

```
4 TOOL CALL "TCH PROBE" Z
5 TCH PROBE 450 SAVE KINEMATICS
    Q410=0    ;MODE
    Q409=5    ;MEMORY
6 TCH PROBE 452 PRESET COMPENSATION
    Q407=12.5 ;SPHERE RADIUS
    Q320=0    ;SET-UP CLEARANCE
    Q408=0    ;RETR. HEIGHT
    Q253=750  ;F PRE-POSITIONING
    Q380=0    ;REFERENCE ANGLE
    Q411=-90  ;START ANGLE A AXIS
    Q412=+90  ;END ANGLE A AXIS
    Q413=0    ;INCID. ANGLE A AXIS
    Q414=0    ;MEAS. POINTS A AXIS
    Q415=-90  ;START ANGLE B AXIS
    Q416=+90  ;END ANGLE B AXIS
    Q417=0    ;INCID. ANGLE B AXIS
    Q418=2    ;MEAS. POINTS B AXIS
    Q419=-90  ;START ANGLE C AXIS
    Q420=+90  ;END ANGLE C AXIS
    Q421=0    ;INCID. ANGLE C AXIS
    Q422=2    ;MEAS. POINTS C AXIS
    Q423=4    ;NO. OF MEAS. POINTS
    Q432=0    ;BACKLASH, ANG. RANGE
```

- **End angle B axis** Q416 (absolute): Ending angle in the B axis at which the last measurement is to be made. Input range -359.999 to 359.999
- **Angle of incid. in B axis** Q417: Angle of incidence in the B axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999
- **Number meas. points B axis** Q418: Number of probe measurements with which the TNC is to measure the B axis. If the input value = 0, the TNC does not measure the respective axis. Input range 0 to 12
- **Start angle C axis** Q419 (absolute): Starting angle in the C axis at which the first measurement is to be made. Input range -359.999 to 359.999
- **End angle C axis** Q420 (absolute): Ending angle in the C axis at which the last measurement is to be made. Input range -359.999 to 359.999
- **Angle of incid. in C axis** Q421: Angle of incidence in the C axis at which the other rotary axes are to be measured. Input range -359.999 to 359.999
- **Number meas. points C axis** Q422: Number of probe measurements with which the TNC is to measure the C axis. If the input value = 0, the TNC does not measure the respective axis. Input range 0 to 12
- **No. of measuring points** Q423: Specify the number of probing points to be used by the TNC for measuring the calibration sphere in the plane. Input range: 3 to 8 measurements
- **Backlash, angle range** Q432: Here you define the angle value to be used as traverse for the measurement of the rotary axis backlash. The traversing angle must be significantly larger than the actual backlash of the rotary axes. If input value = 0, the TNC does not measure the backlash. Input range -3.0000 to +3.0000

Adjustment of interchangeable heads

The goal of this procedure is for the workpiece preset to remain unchanged after changing rotary axes (head exchange).

In the following example, a fork head is adjusted to the A and C axes. The A axis is changed, whereas the C axis continues being a part of the basic configuration.

- Insert the interchangeable head that will be used as a reference head
- Clamp the calibration sphere
- Insert the touch probe
- Use Cycle 451 to measure the complete kinematics, including the reference head
- Set the preset (using Q431 = 2 or 3 in Cycle 451) after measuring the reference head

Example: Measuring a reference head

```
1 TOOL CALL "TCH PROBE" Z
2 TCH PROBE 451 MEASURE KINEMATICS
    Q406=1    ;MODE
    Q407=12.5 ;SPHERE RADIUS
    Q320=0    ;SET-UP CLEARANCE
    Q408=0    ;RETR. HEIGHT
    Q253=2000 ;F PRE-POSITIONING
    Q380=45   ;REFERENCE ANGLE
    Q411=-90  ;START ANGLE A AXIS
    Q412=+90  ;END ANGLE A AXIS
    Q413=45   ;INCID. ANGLE A AXIS
    Q414=4    ;MEAS. POINTS A AXIS
    Q415=-90  ;START ANGLE B AXIS
    Q416=+90  ;END ANGLE B AXIS
    Q417=0    ;INCID. ANGLE B AXIS
    Q418=2    ;MEAS. POINTS B AXIS
    Q419=+90  ;START ANGLE C AXIS
    Q420=+270 ;END ANGLE C AXIS
    Q421=0    ;INCID. ANGLE C AXIS
    Q422=3    ;MEAS. POINTS C AXIS
    Q423=4    ;NO. OF MEAS. POINTS
    Q431=3    ;PRESET
    Q432=0    ;BACKLASH, ANG. RANGE
```

- Insert the second interchangeable head
- Insert the touch probe
- Measure the interchangeable head with Cycle 452
- Measure only the axes that have actually been changed (in this example: only the A axis; the C axis is hidden with Q422)
- The preset and the position of the calibration sphere must not be changed during the complete process
- All other interchangeable heads can be adjusted in the same way

The head change function can vary depending on the individual machine tool. Refer to your machine manual.

Example: Adjusting an interchangeable head

```
3 TOOL CALL "TCH PROBE" Z
4 TCH PROBE 452 PRESET COMPENSATION
    Q407=12.5 ;SPHERE RADIUS
    Q320=0    ;SET-UP CLEARANCE
    Q408=0    ;RETR. HEIGHT
    Q253=2000 ;F PRE-POSITIONING
    Q380=45   ;REFERENCE ANGLE
    Q411=-90  ;START ANGLE A AXIS
    Q412=+90  ;END ANGLE A AXIS
    Q413=45   ;INCID. ANGLE A AXIS
    Q414=4    ;MEAS. POINTS A AXIS
    Q415=-90  ;START ANGLE B AXIS
    Q416=+90  ;END ANGLE B AXIS
    Q417=0    ;INCID. ANGLE B AXIS
    Q418=2    ;MEAS. POINTS B AXIS
    Q419=+90  ;START ANGLE C AXIS
    Q420=+270 ;END ANGLE C AXIS
    Q421=0    ;INCID. ANGLE C AXIS
    Q422=0    ;MEAS. POINTS C AXIS
    Q423=4    ;NO. OF MEAS. POINTS
    Q432=0    ;BACKLASH, ANG. RANGE
```

Drift compensation

During machining various machine components are subject to drift due to varying ambient conditions. If the drift remains sufficiently constant over the range of traverse, and if the calibration sphere can be left on the machine table during machining, the drift can be measured and compensated with Cycle 452.

- Clamp the calibration sphere
- Insert the touch probe
- Measure the complete kinematics with Cycle 451 before starting the machining process
- Set the preset (using Q432 = 2 or 3 in Cycle 451) after measuring the kinematics.
- Then set the presets on your workpieces and start the machining process

Example: Reference measurement for drift compensation

```
1 TOOL CALL "TCH PROBE" Z
2 CYCL DEF 247 DATUM SETTING
    Q339=1     ;DATUM NUMBER
3 TCH PROBE 451 MEASURE KINEMATICS
    Q406=1     ;MODE
    Q407=12.5  ;SPHERE RADIUS
    Q320=0     ;SET-UP CLEARANCE
    Q408=0     ;RETR. HEIGHT
    Q253=750   ;F PRE-POSITIONING
    Q380=45    ;REFERENCE ANGLE
    Q411=+90   ;START ANGLE A AXIS
    Q412=+270  ;END ANGLE A AXIS
    Q413=45    ;INCID. ANGLE A AXIS
    Q414=4     ;MEAS. POINTS A AXIS
    Q415=-90   ;START ANGLE B AXIS
    Q416=+90   ;END ANGLE B AXIS
    Q417=0     ;INCID. ANGLE B AXIS
    Q418=2     ;MEAS. POINTS B AXIS
    Q419=+90   ;START ANGLE C AXIS
    Q420=+270  ;END ANGLE C AXIS
    Q421=0     ;INCID. ANGLE C AXIS
    Q422=3     ;MEAS. POINTS C AXIS
    Q423=4     ;NO. OF MEAS. POINTS
    Q431=3     ;PRESET
    Q432=0     ;BACKLASH, ANG. RANGE
```

- Measure the drift of the axes at regular intervals.
- Insert the touch probe.
- Activate the preset in the calibration sphere.
- Use Cycle 452 to measure the kinematics.
- The preset and the position of the calibration sphere must not be changed during the complete process

This procedure can also be performed on machines without rotary axes.

Example: Drift compensation

```
4 TOOL CALL "TCH PROBE" Z
5 TCH PROBE 452 PRESET COMPENSATION
    Q407=12.5 ;SPHERE RADIUS
    Q320=0    ;SET-UP CLEARANCE
    Q408=0    ;RETR. HEIGHT
    Q253=99999;F PRE-POSITIONING
    Q380=45   ;REFERENCE ANGLE
    Q411=-90  ;START ANGLE A AXIS
    Q412=+90  ;END ANGLE A AXIS
    Q413=45   ;INCID. ANGLE A AXIS
    Q414=4    ;MEAS. POINTS A AXIS
    Q415=-90  ;START ANGLE B AXIS
    Q416=+90  ;END ANGLE B AXIS
    Q417=0    ;INCID. ANGLE B AXIS
    Q418=2    ;MEAS. POINTS B AXIS
    Q419=+90  ;START ANGLE C AXIS
    Q420=+270 ;END ANGLE C AXIS
    Q421=0    ;INCID. ANGLE C AXIS
    Q422=3    ;MEAS. POINTS C AXIS
    Q423=3    ;NO. OF MEAS. POINTS
    Q432=0    ;BACKLASH, ANG. RANGE
```

Log function

After running Cycle 452, the TNC creates a measuring log **(TCHPR452.TXT)** containing the following information:

- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Active kinematic number
- Entered calibration sphere radius
- For each measured rotary axis:
  - Starting angle
  - End angle
  - Angle of incidence
  - Number of measuring points
  - Dispersion (standard deviation)
  - Maximum error
  - Angular error
  - Averaged backlash
  - Averaged positioning error
  - Measuring circle radius
  - Compensation values in all axes (preset shift)
  - Evaluation of measuring points
  - Measurement uncertainty of rotary axes

Notes on log data

(see "Notes on log data" on page 498)

19

Touch Probe Cycles: Automatic Tool Measurement

19.1 Fundamentals

Overview

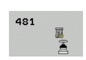
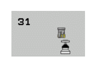
The TNC and the machine tool must be set up by the machine tool builder for use of the TT touch probe.

Some cycles and functions may not be provided on your machine tool. Refer to your machine manual.

In conjunction with the TNC's tool measurement cycles, the tool touch probe enables you to measure tools automatically. The compensation values for tool length and radius can be stored in the central tool file TOOL.T and are accounted for at the end of the touch probe cycle. The following types of tool measurement are provided:

- Tool measurement while the tool is at standstill
- Tool measurement while the tool is rotating
- Measurement of individual teeth

You can program the cycles for tool measurement in the Programming and Editing mode of operation via the TOUCH PROBE key. The following cycles are available:

Cycle	New format	Old format	Page
Calibrating the TT, Cycles 30 and 480			Page 517
Calibrating the wireless TT 449, Cycle 484			Page 519
Measuring the tool length, Cycles 31 and 481			Page 520
Measuring the tool radius, Cycles 32 and 482			Page 522
Measuring the tool length and radius, Cycles 33 and 483			Page 524

The measuring cycles can be used only when the central tool file TOOL.T is active.

Before working with the measuring cycles, you must first enter all the required data into the central tool file and call the tool to be measured with TOOL CALL.

You can also measure tools in a tilted working plane.

Differences between Cycles 31 to 33 and Cycles 481 to 483

The features and the operating sequences are absolutely identical. There are only two differences between Cycles 31 to 33 and Cycles 481 to 483:

- Cycles 481 to 483 are also available in controls for ISO programming under G481 to G483.
- Instead of a selectable parameter for the status of the measurement, the new cycles use the fixed parameter **Q199**.

Setting the machine parameters

The TNC uses the feed rate for probing defined in MP6520 when measuring a tool at standstill.

When measuring a rotating tool, the TNC automatically calculates the spindle speed and feed rate for probing.

The spindle speed is calculated as follows:

n = MP6570 / (r • 0.0063) where

n	Shaft speed [rpm]
MP6570	Maximum permissible cutting speed in m/min
r	Active tool radius in mm

The feed rate for probing is calculated from:

v = meas. tolerance • n where

v	Feed rate for probing in mm/min
Measuring tolerance	Measuring tolerance [mm], depending on MP6507
n	Shaft speed in rpm

MP6507 determines the calculation of the probing feed rate:

MP6507=0:

The measuring tolerance remains constant regardless of the tool radius. With very large tools, however, the feed rate for probing is reduced to zero. The smaller you set the maximum permissible rotational speed (MP6570) and the permissible tolerance (MP6510), the sooner you will encounter this effect.

MP6507=1:

The measuring tolerance is adjusted relative to the size of the tool radius. This ensures a sufficient feed rate for probing even with large tool radii. The TNC adjusts the measuring tolerance according to the following table:

Tool radius	Measuring tolerance
Up to 30 mm	MP6510
30 to 60 mm:	2 • MP6510
60 to 90 mm:	3 • MP6510
90 to 120 mm:	4 • MP6510

MP6507=2:

The feed rate for probing remains constant; the error of measurement, however, rises linearly with the increase in tool radius:

Measuring tolerance = (r • MP6510)/ 5 mm), where

r	Active tool radius in mm
MP6510	Maximum permissible error of measurement

Entries in the tool table TOOL.T

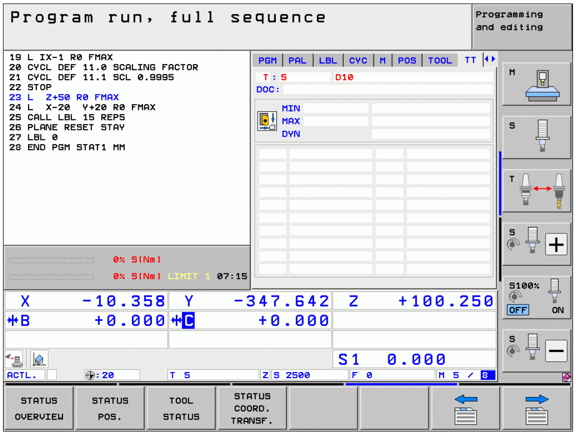
Abbr.	Inputs	Dialog
CUT	Number of teeth (20 teeth maximum)	**Number of teeth?**
LTOL	Permissible deviation from tool length L for wear detection. If the entered value is exceeded, the TNC locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Wear tolerance: length?**
RTOL	Permissible deviation from tool radius R for wear detection. If the entered value is exceeded, the TNC locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Wear tolerance: radius?**
DIRECT.	Cutting direction of the tool for measuring the tool during rotation	**Cutting direction (M3 = –)?**
TT:R-OFFS	Tool length measurement: Tool offset between stylus center and tool center. Preset value: Tool radius R (NO ENT means **R**)	**Tool offset: radius?**
TT:L-OFFS	Radius measurement: Tool offset in addition to MP6530 between upper surface of stylus and lower surface of tool. Default: 0	**Tool offset: length?**
LBREAK	Permissible deviation from tool length L for breakage detection. If the entered value is exceeded, the TNC locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Breakage tolerance: length?**
RBREAK	Permissible deviation from tool radius R for breakage detection. If the entered value is exceeded, the TNC locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Breakage tolerance: radius?**

Input examples for common tool types

Tool type	CUT	TT:R-OFFS	TT:L-OFFS
Drill	– (no function)	0 (no offset required because tool tip is to be measured)	
End mill with diameter < 19 mm	4 (4 teeth)	0 (no offset required because tool diameter is smaller than the contact plate diameter of the TT)	0 (no additional offset required during radius measurement. Offset from MP6530 is used.)
End mill with diameter > 19 mm	4 (4 teeth)	R (offset required because tool diameter is larger than the contact plate diameter of the TT)	0 (no additional offset required during radius measurement. Offset from MP6530 is used.)
Radius cutter	4 (4 teeth)	0 (no offset required because the south pole of the ball is to be measured)	5 (always define the tool radius as the offset so that the diameter is not measured in the radius)

Display of the measurement results

You can display the results of tool measurement in the additional status display (in the machine operating modes). The TNC then shows the program blocks in the left and the measuring results in the right screen window. The measuring results that exceed the permissible wear tolerance are marked in the status display with an asterisk “*”; the results that exceed the permissible breakage tolerance are marked with the character B.



19.2 Calibrating the TT (Cycle 30 or 480, DIN/ISO: G480)

Cycle run

The TT is calibrated with the measuring cycle TCH PROBE 30 or TCH PROBE 480 (see also "Differences between Cycles 31 to 33 and Cycles 481 to 483" on page 513). The calibration process is automatic. The TNC also measures the center misalignment of the calibrating tool automatically by rotating the spindle by 180° after the first half of the calibration cycle.

The calibrating tool must be a precisely cylindrical part, for example a cylinder pin. The resulting calibration values are stored in the TNC memory and are accounted for during subsequent tool measurement.

The calibration tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck. This configuration causes a deformation of 0.1 µm per 1 N of probing force.

Please note while programming:

The functioning of the calibration cycle is dependent on MP6500. Refer to your machine manual.

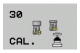
Before calibrating the touch probe, you must enter the exact length and radius of the calibrating tool into the tool table TOOL.T.

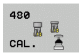
The position of the TT within the machine working space must be defined by setting the Machine Parameters 6580.0 to 6580.2.

If you change the setting of any of the Machine Parameters 6580.0 to 6580.2, you must recalibrate the TT.

During calibration, ensure that no fixtures are attached near or to the touch probe. Recommendation: clear an area at least twice the diameter of the calibration tool

Cycle parameters





- **`Clearance height`**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the calibration tool above the level of the probe contact (safety zone from MP6540). Input range -99999.9999 to 99999.9999; alternatively **`PREDEF`**

Example: NC blocks in old format

```
6 TOOL CALL 1 Z
7 TCH PROBE 30.0 CALIBRATE TT
8 TCH PROBE 30.1 HEIGHT: +90
```

Example: NC blocks in new format

```
6 TOOL CALL 1 Z
7 TCH PROBE 480 CALIBRATE TT
    Q260=+100 ;CLEARANCE HEIGHT
```

19.3 Calibrating the wireless TT 449 (Cycle 484, DIN/ISO: G484)

Fundamentals

With Cycle 484, you calibrate the wireless infrared TT 449 tool touch probe. The calibration process is not completely automated, because the TT's position on the table is not defined.

Cycle run

- Insert the calibrating tool
- Define and start the calibration cycle
- Position the calibrating tool manually above the center of the touch probe and follow the instructions in the pop-up window. Ensure that the calibrating tool is located above the measuring surface of the probe contact

The calibration process is semi-automatic. The TNC also measures the center misalignment of the calibrating tool by rotating the spindle by 180° after the first half of the calibration cycle.

The calibrating tool must be a precisely cylindrical part, for example a cylinder pin. The resulting calibration values are stored in the TNC memory and are accounted for during subsequent tool measurement.

> The calibration tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck. This configuration causes a deformation of 0.1 µm per 1 N of probing force.

Please note while programming:

> The functioning of the calibration cycle is dependent on MP 6500. Refer to your machine manual.
>
> Before calibrating the touch probe, you must enter the exact length and radius of the calibrating tool into the tool table TOOL.T.
>
> The TT needs to be recalibrated if you change its position on the table.

Cycle parameters

Cycle 484 has no cycle parameters.

19.4 Measuring the Tool Length (Cycle 31 or 481, DIN/ISO: G481)

Cycle run

To measure the tool length, program the measuring cycle TCH PROBE 31 or TCH PROBE 481 (see also "Differences between Cycles 31 to 33 and Cycles 481 to 483" on page 513). Via input parameters you can measure the length of a tool by three methods:

- If the tool diameter is larger than the diameter of the measuring surface of the TT, you measure the tool while it is rotating.
- If the tool diameter is smaller than the diameter of the measuring surface of the TT, or if you are measuring the length of a drill or spherical cutter, you measure the tool while it is at standstill.
- If the tool diameter is larger than the diameter of the measuring surface of the TT, you measure the individual teeth of the tool while it is at standstill.

Cycle for measuring a tool during rotation

The control determines the longest tooth of a rotating tool by positioning the tool to be measured at an offset to the center of the touch probe and then moving it toward the measuring surface of the TT until it contacts the surface. The offset is programmed in the tool table under Tool offset: Radius (**TT: R-OFFS**).

Cycle for measuring a tool during standstill (e.g. for drills)

The control positions the tool to be measured over the center of the measuring surface. It then moves the non-rotating tool toward the measuring surface of the TT until contact is made. To activate this function, enter zero for the tool offset: Radius (**TT: R-OFFS**) in the tool table.

Cycle for measuring individual teeth

The TNC pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the tip of the tool to the upper edge of the touch probe head is defined in MP6530. You can enter an additional offset with tool offset: Length (**TT: L-OFFS**) in the tool table. The TNC probes the tool radially during rotation to determine the starting angle for measuring the individual teeth. It then measures the length of each tooth by changing the corresponding angle of spindle orientation. To activate this function, program TCH PROBE 31 = 1 for CUTTER MEASUREMENT.

Please note while programming:

Before measuring a tool for the first time, enter the following data on the tool into the tool table TOOL.T: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

You can run an individual tooth measurement of tools with up to **99 teeth**. The TNC shows the measured values of up to 24 teeth in the status display.

Cycle parameters

- **Measure tool=0 / Check tool=1**: Select whether the tool is to be measured for the first time or whether a tool that has already been measured is to be inspected. If the tool is being measured for the first time, the TNC overwrites the tool length L in the central tool file TOOL.T by the delta value DL = 0. If you wish to inspect a tool, the TNC compares the measured length with the tool length L that is stored in TOOL.T. It then calculates the positive or negative deviation from the stored value and enters it into TOOL.T as the delta value DL. The deviation can also be used for parameter Q115. If the delta value is greater than the permissible tool length tolerance for wear or break detection, the TNC will lock the tool (status L in TOOL.T).
- **Parameter number for result?**: Parameter number in which the TNC stores the status of the measurement:
  0.0: Tool is within the tolerance
  1.0: Tool is worn (**LTOL** exceeded)
  2.0: Tool is broken (**LBREAK** exceeded). If you do not wish to use the result of measurement within the program, answer the dialog prompt with NO ENT.
- **Clearance height**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from MP6540). Input range -99999.9999 to 99999.9999; alternatively **PREDEF**
- **Cutter measurement? 0=No / 1=Yes**: Choose whether the control is to measure the individual teeth (maximum of 99 teeth)

Example: Measuring a rotating tool for the first time; old format

```
6 TOOL CALL 12 Z
7 TCH PROBE 31.0 TOOL LENGTH
8 TCH PROBE 31.1 CHECK: 0
9 TCH PROBE 31.2 HEIGHT: +120
10 TCH PROBE 31.3 PROBING THE TEETH: 0
```

Example: Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
6 TOOL CALL 12 Z
7 TCH PROBE 31.0 TOOL LENGTH
8 TCH PROBE 31.1 CHECK: 1 Q5
9 TCH PROBE 31.2 HEIGHT: +120
10 TCH PROBE 31.3 PROBING THE TEETH: 1
```

Example: NC blocks in new format

```
6 TOOL CALL 12 Z
7 TCH PROBE 481 TOOL LENGTH
    Q340=1    ;CHECK
    Q260=+100 ;CLEARANCE HEIGHT
    Q341=1    ;PROBING THE TEETH
```

19.5 Measuring the Tool Radius (Cycle 32 or 482, DIN/ISO: G482)

Cycle run

To measure the tool radius, program the cycle TCH PROBE 32 or TCH PROBE 482 (see also "Differences between Cycles 31 to 33 and Cycles 481 to 483" on page 513). Select via input parameters by which of two methods the radius of a tool is to be measured:

- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth.

The TNC pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the tip of the milling tool to the upper edge of the touch probe head is defined in MP6530. The TNC probes the tool radially while it is rotating. If you have programmed a subsequent measurement of individual teeth, the control measures the radius of each tooth with the aid of oriented spindle stops.

Please note while programming:

Before measuring a tool for the first time, enter the following data on the tool into the tool table TOOL.T: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

Cylindrical tools with diamond surfaces can be measured with stationary spindle. To do so, define in the tool table the number of teeth (CUT) as 0 and adjust machine parameter 6500. Refer to your machine manual.

You can run an individual tooth measurement of tools with up to **99 teeth**. The TNC shows the measured values of up to 24 teeth in the status display.

Cycle parameters

- **`Measure tool=0 / Check tool=1`**: Select whether the tool is to be measured for the first time or whether a tool that has already been measured is to be inspected. If the tool is being measured for the first time, the TNC overwrites the tool radius R in the central tool file TOOL.T by the delta value DR = 0. If you wish to inspect a tool, the TNC compares the measured radius with the tool radius R that is stored in TOOL.T. It then calculates the positive or negative deviation from the stored value and enters it into TOOL.T as the delta value DR. The deviation can also be used for parameter Q116. If the delta value is greater than the permissible tool radius tolerance for wear or break detection, the TNC will lock the tool (status L in TOOL.T).
- **`Parameter number for result?`**: Parameter number in which the TNC stores the status of the measurement:
  0.0: Tool is within the tolerance
  1.0: Tool is worn (**`RTOL`** exceeded)
  2.0: Tool is broken (**`RBREAK`** exceeded). If you do not wish to use the result of measurement within the program, answer the dialog prompt with NO ENT.
- **`Clearance height`**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from MP6540). Input range -99999.9999 to 99999.9999; alternatively **`PREDEF`**
- **`Cutter measurement? 0=No / 1=Yes`**: Choose whether the control is also to measure the individual teeth (maximum of 99 teeth)

Example: Measuring a rotating tool for the first time; old format

```
6 TOOL CALL 12 Z
7 TCH PROBE 32.0 TOOL RADIUS
8 TCH PROBE 32.1 CHECK: 0
9 TCH PROBE 32.2 HEIGHT: +120
10 TCH PROBE 32.3 PROBING THE TEETH: 0
```

Example: Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
6 TOOL CALL 12 Z
7 TCH PROBE 32.0 TOOL RADIUS
8 TCH PROBE 32.1 CHECK: 1 Q5
9 TCH PROBE 32.2 HEIGHT: +120
10 TCH PROBE 32.3 PROBING THE TEETH: 1
```

Example: NC blocks in new format

```
6 TOOL CALL 12 Z
7 TCH PROBE 482 TOOL RADIUS
    Q340=1    ;CHECK
    Q260=+100 ;CLEARANCE HEIGHT
    Q341=1    ;PROBING THE TEETH
```

19.6 Measuring Tool Length and Radius (Cycle 33 or 483, DIN/ISO: G483)

Cycle run

To measure both the length and radius of a tool, program the measuring cycle TCH PROBE 33 or TCH PROBE 483 (see also "Differences between Cycles 31 to 33 and Cycles 481 to 483" on page 513). This cycle is particularly suitable for the first measurement of tools, as it saves time when compared with individual measurement of length and radius. Via input parameters you can select the desired type of measurement:

- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth.

The TNC measures the tool in a fixed programmed sequence. First it measures the tool radius, then the tool length. The sequence of measurement is the same as for measuring cycles 31 and 32.

Please note while programming:

Before measuring a tool for the first time, enter the following data on the tool into the tool table TOOL.T: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

Cylindrical tools with diamond surfaces can be measured with stationary spindle. To do so, define in the tool table the number of teeth (CUT) as 0 and adjust machine parameter 6500. Refer to your machine manual.

You can run an individual tooth measurement of tools with up to **99 teeth**. The TNC shows the measured values of up to 24 teeth in the status display.

Cycle parameters

- **`Measure tool=0 / Check tool=1`**: Select whether the tool is to be measured for the first time or whether a tool that has already been measured is to be inspected. If the tool is being measured for the first time, the TNC overwrites the tool radius R and the tool length L in the central tool file TOOL.T and sets the delta values DR and DL = 0. If you are checking a tool, the measured tool data is compared with the data in the tool table. The TNC calculates the deviations and enters them as positive or negative delta values DR and DL in TOOL.T. The deviations can also be used for parameters Q115 and Q116. If the delta values are greater than the permissible tool tolerances for wear or break detection, the TNC will lock the tool (status L in TOOL.T).
- **`Parameter number for result?`**: Parameter number in which the TNC stores the status of the measurement:
  0.0: Tool is within the tolerance
  1.0: Tool is worn (**`LTOL`** or/and **`RTOL`** exceeded)
  2.0: Tool is broken (**`LBREAK`** or/and **`RBREAK`** exceeded). If you do not wish to use the result of measurement within the program, answer the dialog prompt with NO ENT.
- **`Clearance height`**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from MP6540). Input range -99999.9999 to 99999.9999; alternatively **`PREDEF`**
- **`Cutter measurement? 0=No / 1=Yes`**: Choose whether the control is also to measure the individual teeth (maximum of 99 teeth)

Example: Measuring a rotating tool for the first time; old format

```
6 TOOL CALL 12 Z
7 TCH PROBE 33.0 MEASURE TOOL
8 TCH PROBE 33.1 CHECK: 0
9 TCH PROBE 33.2 HEIGHT: +120
10 TCH PROBE 33.3 PROBING THE TEETH: 0
```

Example: Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
6 TOOL CALL 12 Z
7 TCH PROBE 33.0 MEASURE TOOL
8 TCH PROBE 33.1 CHECK: 1 Q5
9 TCH PROBE 33.2 HEIGHT: +120
10 TCH PROBE 33.3 PROBING THE TEETH: 1
```

Example: NC blocks in new format

```
6 TOOL CALL 12 Z
7 TCH PROBE 483 MEASURE TOOL
    Q340=1    ;CHECK
    Q260=+100 ;CLEARANCE HEIGHT
    Q341=1    ;PROBING THE TEETH
```

Overview

Fixed cycles

Cycle number	Cycle designation	DEF active	CALL active	Page
7	Datum shift	■		Page 279
8	Mirroring	■		Page 287
9	Dwell time	■		Page 309
10	Rotation	■		Page 289
11	Scaling factor	■		Page 291
12	Program call	■		Page 310
13	Oriented spindle stop	■		Page 312
14	Contour definition	■		Page 189
19	Tilting the working plane	■		Page 295
20	SL II contour data	■		Page 194
21	SL II pilot drilling		■	Page 196
22	SL II rough out		■	Page 198
23	Floor finishing SL II		■	Page 202
24	Side finishing SL II		■	Page 203
25	Contour train		■	Page 207
26	Axis-specific scaling	■		Page 293
27	Cylinder surface		■	Page 227
28	Cylindrical surface slot		■	Page 230
29	Cylinder surface ridge		■	Page 233
30	Run 3-D data		■	Page 261
32	Tolerance	■		Page 313
39	Cylinder surface external contour		■	Page 236
200	Drilling		■	Page 79
201	Reaming		■	Page 81
202	Boring		■	Page 83
203	Universal drilling		■	Page 87

Cycle number	Cycle designation	DEF active	CALL active	Page
204	Back boring		■	Page 91
205	Universal pecking		■	Page 95
206	Tapping with a floating tap holder, new		■	Page 111
207	Rigid tapping, new		■	Page 113
208	Bore milling		■	Page 99
209	Tapping with chip breaking		■	Page 116
220	Polar pattern	■		Page 177
221	Cartesian pattern	■		Page 180
230	Multipass milling		■	Page 263
231	Ruled surface		■	Page 265
232	Face milling		■	Page 269
240	Centering		■	Page 77
241	Single-fluted deep-hole drilling		■	Page 102
247	Datum setting	■		Page 286
251	Rectangular pocket (complete machining)		■	Page 145
252	Circular pocket (complete machining)		■	Page 150
253	Slot milling		■	Page 154
254	Circular slot		■	Page 159
256	Rectangular stud (complete machining)		■	Page 164
257	Circular stud (complete machining)		■	Page 168
262	Thread milling		■	Page 121
263	Thread milling/countersinking		■	Page 124
264	Thread drilling/milling		■	Page 128
265	Helical thread drilling/milling		■	Page 132
267	Outside thread milling		■	Page 136
270	Contour train data	■		Page 205
275	Trochoidal slot		■	Page 209
290	Interpolation turning		■	Page 323

Touch probe cycles

Cycle number	Cycle designation	DEF active	CALL active	Page
0	Reference plane	■		Page 416
1	Polar datum	■		Page 417
2	Calibrate TS radius	■		Page 461
3	Measuring	■		Page 463
4	Measuring in 3-D	■		Page 465
9	Calibrate TS length	■		Page 462
30	Calibrate the TT	■		Page 513
31	Measure/Inspect the tool length	■		Page 515
32	Measure/Inspect the tool radius	■		Page 517
33	Measure/Inspect the tool length and the tool radius	■		Page 519
400	Basic rotation using two points	■		Page 336
401	Basic rotation from two holes	■		Page 339
402	Basic rotation from two studs	■		Page 342
403	Compensate misalignment with rotary axis	■		Page 345
404	Set basic rotation	■		Page 349
405	Compensate misalignment with the C axis	■		Page 350
408	Reference point at slot center (FCL 3 function)	■		Page 359
409	Reference point at ridge center (FCL 3 function)	■		Page 363
410	Datum from inside of rectangle	■		Page 366
411	Datum from outside of rectangle	■		Page 370
412	Datum from inside of circle (hole)	■		Page 374
413	Datum from outside of circle (stud)	■		Page 378
414	Datum from outside of corner	■		Page 382
415	Datum from inside of corner	■		Page 387
416	Datum from circle center	■		Page 391
417	Datum in touch probe axis	■		Page 395
418	Datum at center between four holes	■		Page 397
419	Datum in any one axis	■		Page 401

Cycle number	Cycle designation	DEF active	CALL active	Page
420	Workpiece—measure angle	■		Page 419
421	Workpiece—measure hole (center and diameter of hole)	■		Page 422
422	Workpiece—measure circle from outside (diameter of circular stud)	■		Page 426
423	Workpiece—measure rectangle from inside	■		Page 430
424	Workpiece—measure rectangle from outside	■		Page 434
425	Workpiece—measure inside width (slot)	■		Page 438
426	Workpiece—measure outside width (ridge)	■		Page 441
427	Workpiece—measure in any selectable axis	■		Page 444
430	Workpiece—measure bolt hole circle	■		Page 447
431	Workpiece—measure plane	■		Page 451
440	Measure axis shift	■		Page 467
441	Rapid probing: Set global touch probe parameters (FCL 2 function)	■		Page 470
450	KinematicsOpt: Save kinematics (option)	■		Page 478
451	KinematicsOpt: Measure kinematics (option)	■		Page 480
452	KinematicsOpt: Preset compensation (option)	■		Page 480
460	Calibrate TS: Radius and length calibration on a calibration sphere	■		Page 472
480	Calibrate the TT	■		Page 513
481	Measure/Inspect the tool length	■		Page 515
482	Measure/Inspect the tool radius	■		Page 517
483	Measure/Inspect the tool length and the tool radius	■		Page 519
484	Calibrate infrared TT	■		Page 514

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 8669 31-0
FAX +49 8669 32-5061
E-mail: info@heidenhain.de

Technical support FAX +49 8669 32-1000
Measuring systems ☏ +49 8669 31-3104
E-mail: service.ms-support@heidenhain.de
TNC support ☏ +49 8669 31-3101
E-mail: service.nc-support@heidenhain.de
NC programming ☏ +49 8669 31-3103
E-mail: service.nc-pgm@heidenhain.de
PLC programming ☏ +49 8669 31-3102
E-mail: service.plc@heidenhain.de
Lathe controls ☏ +49 8669 31-3105
E-mail: service.lathe-support@heidenhain.de

www.heidenhain.de

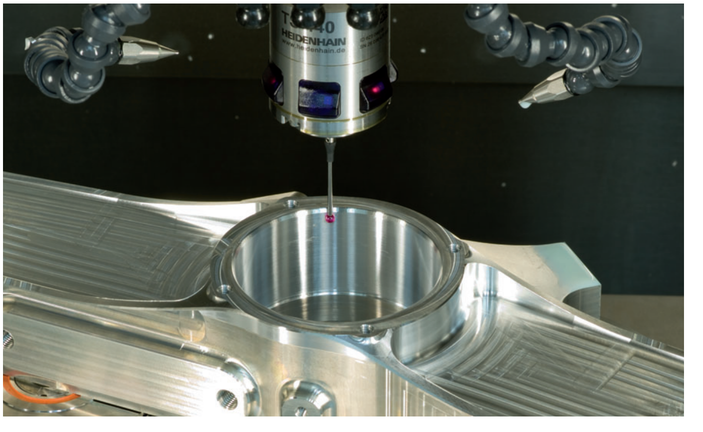
Touch probes from HEIDENHAIN

help you reduce non-productive time and
improve the dimensional accuracy of the finished workpieces.

Workpiece touch probes

TS 220	Signal transmission by cable
TS 440, TS 444	Infrared transmission
TS 640, TS 740	Infrared transmission

- Workpiece alignment
- Setting datums
- Workpiece measurement



Tool touch probes

TT 140	Signal transmission by cable
TT 449	Infrared transmission
TL	Contact-free laser systems

- Tool measurement
- Wear monitoring
- Tool breakage detection

670388-25 · Ver05 · SW04 · 3/2016 · Printed in Germany · F&W